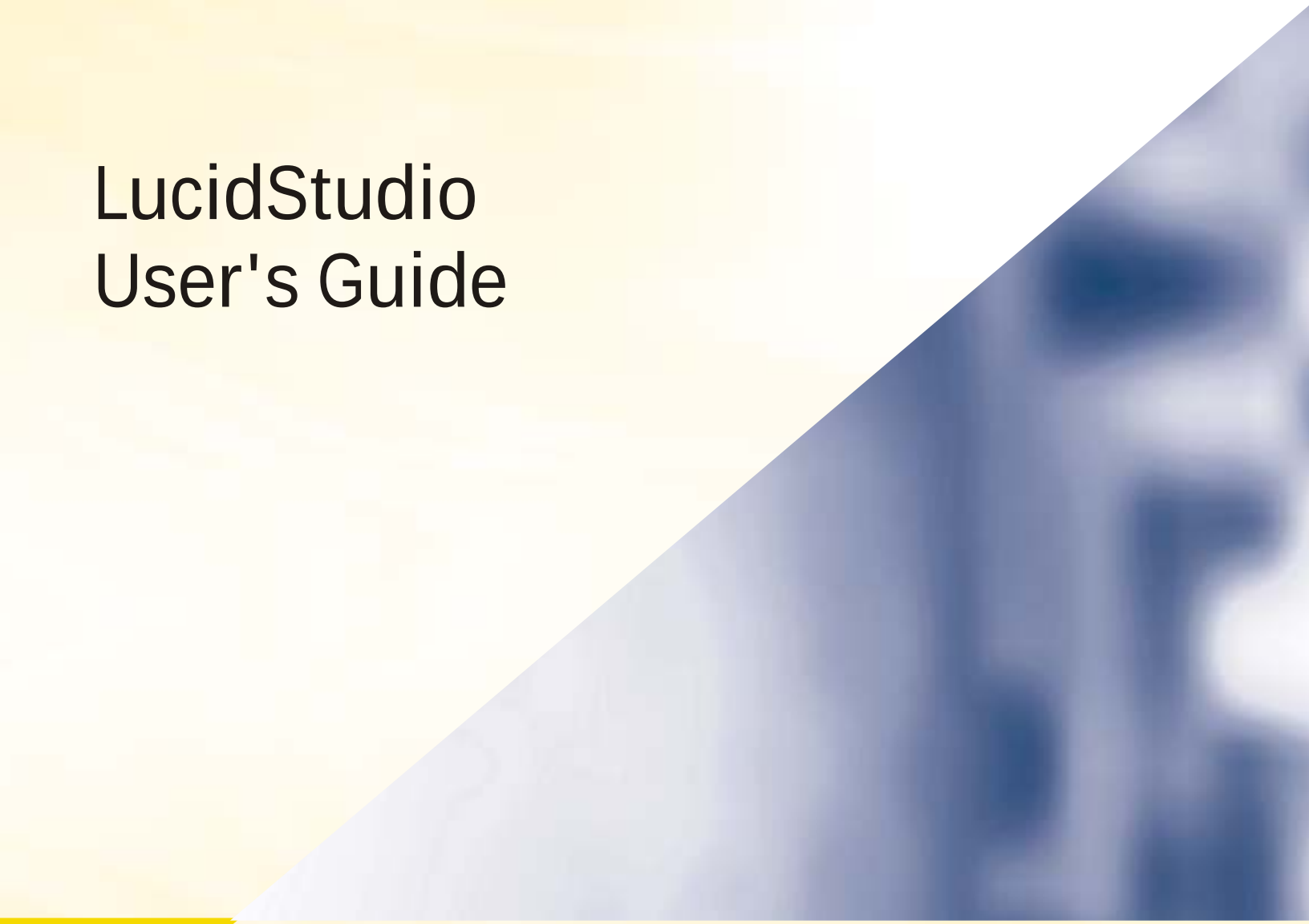
brandenburg gmbh
Software-Lösungen für die Produktentwicklung

LucidStudio User's Guide



LucidShape
Computer Aided Lighting

LucidStudio User's Guide

Version 1.12.1

Published July 08, 2014

Table of Contents

1. Getting started with LucidStudio 11
    1.1. LucidStudio Overview 11
    1.2. Principles of LucidStudio 11
        1.2.1. Shapes 13
        1.2.2. Views 14
        1.2.3. Mouse Interaction 18
        1.2.4. Peculiarities of LucidStudio 20
    1.3. The First Example: A Guide to LucidStudio 21
        1.3.1. Create an Empty Geometry File 21
        1.3.2. Create a Cylinder Light Source 22
        1.3.3. Define a Reflector Surface 24
        1.3.4. Create Sensors 25
        1.3.5. Simulate the scene 27
2. File and Edit 31
    2.1. Editing LucidShape Documents 31
    2.2. Saving LucidShape Documents 31
    2.3. Transferring Geometry between LucidShape and CAD tools 32
        2.3.1. The CATIA Transfer Dialog 33
        2.3.2. The Rhino Transfer Dialog 34
    2.4. Transformed Input and Output 35
    2.5. Menu Operations 35
    2.6. Printing 36
    2.7. Standard Editing 37
3. Views 39
    3.1. GeoView 40
        3.1.1. General Commands 40
        3.1.2. Selection Modes 40
        3.1.3. Viewing Perspectives 40
        3.1.4. Clear 41
        3.1.5. Axis 41
        3.1.6. Objects Coloring and Display 41
        3.1.7. Advanced Display Properties 42
    3.2. Tree View 46
        3.2.1. Selections 46
        3.2.2. Tree Node Properties 46
        3.2.3. Tree Display 47
        3.2.4. Elements Display Filtering 47
    3.3. Text View 47
    3.4. UV View 48
        3.4.1. General Commands 48
        3.4.2. Color Scaling and Scene Lines 48
        3.4.3. Operations on the Data 49
        3.4.4. Background Colors 61
        3.4.5. Spectral specific options 62
        3.4.6. Detailed View Properties 62
    3.5. Light Screen View 66
        3.5.1. General Commands 66
        3.5.2. Display Commands 66
        3.5.3. Coloring and Display Properties 67

3.5.4. Screen View Properties 67
4. Geometry 69
4.1. Basic Operations on Geometric Objects 70
4.1.1. Selection, Deletion and Renaming 70
4.1.2. Transform 71
4.1.3. Set Global Axis 73
4.1.4. Conversion of Curves and Surfaces 73
4.1.5. Modify Data Structures of Geometric Objects 73
4.2. Standard Geometries 75
4.2.1. Rectangular Paraboloid 77
4.2.2. Rotational Paraboloid 77
4.2.3. Rotational Hyperboloid 77
4.2.4. Rotational Ellipsoid 78
4.2.5. Compound Parabolic Concentrator 78
4.2.6. Cylinder 79
4.2.7. Sphere 79
4.2.8. Mesh Sphere 79
4.2.9. Plane 80
4.2.10. Disk 80
4.2.11. Cone 80
4.2.12. Box 81
4.3. Procedural Surfaces 81
4.3.1. Rotational Surface 81
4.3.2. Pipe Surface 82
4.3.3. Torus 82
4.3.4. Coil 82
4.3.5. Extruded Surface 83
4.3.6. Offset Surface 83
4.3.7. Swept Surface 84
4.3.8. Swung Surface 84
4.3.9. Surface from Formula 84
4.4. Standard Curves 86
4.4.1. Parabola 86
4.4.2. Hyperbola 87
4.4.3. Ellipse 87
4.4.4. Ellipse by Two Focal Points 87
4.4.5. Helix 88
4.4.6. Compound Parabolic Concentrator 88
4.4.7. Curve from Formula 88
4.4.8. Curve between two Curves 90
4.4.9. Curves from NURBS Surface Border 90
4.4.10. Curves from Trimmed Surface Border 90
4.4.11. Create Point 90
4.4.12. Create Polyline 91
4.4.13. Create NURBS 92
4.4.14. Create Control Curve 93
4.5. Standard Surfaces 94
4.5.1. Surface from Boundary Layers 94
4.5.2. Surface between Curve and Point 95
4.5.3. Surface between two Curves 95
4.5.4. Surface between two Surfaces 95

4.6. Lenses 96
4.6.1. Aspheric Lens 96
4.6.2. Aspheric Lens Per Formula 97
4.6.3. Free Form Fresnel Lens 98
4.6.4. TIR (Total Internal Reflection) Lens 100
4.6.5. Potato Chip Lens 102
4.6.6. Free Form Variable Fresnel Lens 102
4.6.7. Elliptic Lens 103
4.6.8. Hyperbolic Lens 104
4.6.9. Spherical Lens 105
4.7. Approximated and Interpolated Curve 105
4.7.1. Read Point Row 106
4.7.2. Create Fitting Curve 106
4.7.3. Read Value Row 107
4.7.4. Create Explicit Curve 107
4.8. Approximated and Interpolated Surface 108
4.8.1. Common Dialog Parts 108
4.8.2. Read Point Cloud 109
4.8.3. Create Point Cloud 109
4.8.4. Sample Point Cloud from Target Scene 110
4.8.5. Create Surface from Point Cloud 111
4.8.6. Read Point Grid 112
4.8.7. Create Point Grid 112
4.8.8. Extrapolate Point Grid 113
4.8.9. Sample Point Grid from Target Scene 114
4.8.10. Create Point Grid on Surface 115
4.8.11. Create Fitting Surface 115
4.8.12. Read Value Grid 115
4.8.13. Create Explicit Surface 116
4.8.14. Create Skinned Surface 116
4.8.15. Create Skinned Surface Loft 116
4.8.16. Create Skinned Surface Horizontal/Vertical 117
4.9. Ray Deviation Correction 118
4.10. Various Geometry Tools 119
4.10.1. Create Axis System 119
4.10.2. Create Clipping Plane 120
4.10.3. Create Object Vector 120
4.10.4. Create Polylines from Rays 121
4.10.5. Create Cylinder from Rays 121
4.10.6. Create Annotation Text 121
4.10.7. Create Annotation Bitmap 122
5. FreeForm Geometries Menu Selection 123
5.1. The menu entries for FunGeo 123
5.2. MF Reflector/Lens Surface 124
5.3. Switch Board 130
5.4. PCS Reflector/Lens Surface 132
5.5. PCS Lens 137
5.6. Procedural Surface (PS) Reflector 138
5.7. Procedural Surface (PS) Refractor 141
5.8. Collimator LED Lens 143
5.9. Prismband for Lightguides 149

5.10. Dotmask for Backlighting 153
5.11. Retro Reflector 156
5.12. Genetic Algorithm Optimizer 158
5.13. Model Note 159
5.14. Model Documentation 159
6. Light Sources 163
6.1. Base Options and Parameters 164
6.2. Classic Light Sources 166
6.2.1. Cylinder Light Source 166
6.2.2. Disk Light Source 167
6.2.3. Plane Light Source 167
6.2.4. U-Shape Light Source 168
6.2.5. Coil Shape Light Source 168
6.2.6. RayFile Light Source 169
6.2.7. Point Light Source 170
6.3. Special Light Sources 170
6.3.1. Volume Light Source 170
6.3.2. LED Point Light Source 171
6.3.3. Low Beam, High Beam, Signal Lamp and special Light Sources 172
6.4. Sky Light Source 172
6.5. Ray Trace Bundles 174
6.5.1. Single Ray Bundle 174
6.5.2. Cylinder Filament Bundle 174
6.5.3. Free Surface Filament Bundle 175
6.5.4. Cone Bundle 176
6.5.5. Parallel Bundle 176
6.5.6. Point Set Bundle 177
6.5.7. Parallel On Normal Bundle 177
6.5.8. Tip Cone On Normal Bundle 178
6.5.9. RayFile Bundle 178
6.6. RayFile and RayFile Tools 179
6.6.1. The use of a RayFile 179
6.6.2. Simulation with more than one RayFile 180
6.6.3. Copy RayFile 180
6.6.4. Check RayFile 182
6.6.5. Merge RayFiles 182
6.6.6. Compute a RayFile's Pseudo Focal Point (PFP) 183
6.6.7. Project RayFile on Geometry 184
6.6.8. Lamp Models for RayFiles 184
7. Emitter, Actor and Sensor Materials 187
7.1. The Dialogs Create Material and Assign Material 187
7.2. Emitter Materials 189
7.2.1. Lambertian Emitter 190
7.2.2. Directional Emitter 190
7.2.3. Isoradiant Emitter 191
7.2.4. Ray File Emitter 191
7.2.5. LID File Emitter 191
7.2.6. LID Curve Emitter 191
7.2.7. LID Surface Emitter 192
7.2.8. Spectral Power Distribution (SPD) 192
7.3. Actor Materials 194

7.3.1. Ideal Mirror (Reflective) 198
7.3.2. Ideal Lens (Refractive) 199
7.3.3. Lambertian (Reflective and Refractive) 199
7.3.4. Gaussian (Reflective and Refractive) 200
7.3.5. Gaussian Retro-reflex (Reflective) 200
7.3.6. Lambert/Gauss Combination (Reflective and Refractive) 200
7.3.7. 2D BSDF Theta Curve (Reflective and Refractive) 201
7.3.8. 3D BSDF data from file (Reflective and Refractive) 201
7.3.9. ABg (Reflective) 201
7.3.10. Absorber 202
7.4. Sensors and Sensor Materials 202
7.4.1. Candela Sensor 204
7.4.2. Ray File Sensor 206
7.4.3. Ray History Sensor 206
7.4.4. Volume Sensor 207
7.4.5. Light Flow Sensor 208
7.4.6. Exit Sensors 210
7.4.7. Illumination (Lux), Light Flux (Lumen) and Luminance (Candela per m2) Sensor Materials 210
7.4.8. Luminance Camera 212
7.4.9. License Plate Sensor 215
7.5. Media for Refractive Materials 217
7.6. Spectral Reflection, Refraction and Absorption Distributions 219
7.6.1. User Defined Spectral Distributions 219
7.6.2. Special Dispersion and Emission Functions 220
7.7. The BSDF Wizard 224
7.7.1. BSDFs in a nutshell 224
7.7.2. The Wizard's Dialog Elements 225
7.8. Importing other BSDF Data 227
7.9. Set Colour 228
7.10. Set Texture 228
8. Simulation 231
8.1. Monte Carlo Ray Trace 231
8.2. GPU Trace Simulation 235
8.3. Light Mapping Simulation 236
8.4. Simulate by Spooler 237
8.5. Simulation Manager 239
8.6. Sensor Light Calculations 241
8.6.1. Gather Sensor Light 241
8.6.2. Reverse Sensor Light 241
8.6.3. Luminance Image Sensor Light 242
8.7. Various Commands 244
8.7.1. Random Rays 244
8.7.2. Interrupt 244
9. Analysis of Geometry and LID Data 245
9.1. LID Measuring 245
9.1.1. Apply Test Tables 245
9.1.2. Apply Last Selected Test Table 248
9.1.3. Measure Point or Zone 248
9.2. LID Modeling 249
9.2.1. Trim Cutoff Lines 249

9.2.2. Interpolate LIDs from Test Tables 250
9.3. LID Data Conversion 251
9.3.1. Surface from LID Data 251
9.3.2. Section Curve through LID Data 253
9.4. Analysis of 3D Geometry 255
9.4.1. Bounding Box 255
9.4.2. Visualize Volume Sensor 255
9.4.3. Restore Rays from Ray History Sensor 257
9.4.4. Ray Deviations and Checkerboard Images 260
9.4.5. Wall Thickness Diagram 262
9.5. Benchmark test tool for TC4-45 or self defined assessments 264
9.6. Bitmap Tools 267
9.6.1. Bitmap Show 267
9.6.2. 3D Snapshot 268
10. LIDEditor 269
10.1. Overview 269
10.2. LID Editor Gallery 269
10.3. LID Editor 270
10.3.1. Loading LIDs 270
10.3.2. Options and Parameters 270
10.3.3. Editing LIDs 272
10.4. Dynamic LID Editor 274
10.4.1. Introduction and Principle 274
10.4.2. Example 275
11. Applications 277
11.1. General Automotive Application 277
11.1.1. Static Road Simulation 277
11.1.2. Virtual Beam Pattern 282
11.1.3. License Plate 283
11.1.4. Retro Reflector 284
11.2. General Application 285
11.2.1. Aircraft Lighting 285
11.2.2. Test Room for Indoor Lighting 286
11.2.3. Street Illumination 287
11.2.4. Reverse Street Light Pattern Calculation 294
11.3. Tests 296
11.3.1. BSDF Maker 296
12. Miscellaneous Menus 299
12.1. Custom 299
12.2. Options 299
12.2.1. Global Settings 299
12.2.2. Working Directory for Open/Save 301
12.2.3. Editing of LID Test Tables 302
12.2.4. Hardware Graphics Accelerator 302
12.2.5. COM Server 302
12.3. Window 303
12.4. Help 303
12.4.1. License Check 304
12.4.2. Shortcuts to LucidShape Documents 304
A. LucidShape File Types 305
B. List of Toolbar Icons 307

B.1. Common Tools 307
B.2. View Options and Manipulation Tools 307
B.3. Simulation and Script Execution Tools 307
B.4. Geometry View Tools 308
B.5. UV/Light Data View Tools 308
C. Mouse Interaction Reference 309
C.1. Geometry View 309
C.2. Tree View 309
C.3. Light Screen View 309
C.4. UV/Lightdata View 310
C.5. Text Edit View 310
C.6. Message Box 310
D. Efficacy Scale Factor and Rhino CAD Im-/Export 311
D.1. Efficacy Scale Factor 311
D.2. Materials Import/Export: Rhino CAD 312
Index 313

Chapter 1. Getting started with LucidStudio

1.1. LucidStudio Overview

LucidStudio is the interactive interface to the LucidShape design system. Its graphical user interface follows Microsoft Windows® standard and is thus intuitive, user friendly, and easy to learn and master. It allows all design tasks to be performed at a mouse click.

State of the art computer graphics (OpenGL) are used for the 3D scene views, allowing an easy and fast navigation within the scene. Tree views provide an overview of the light simulation scene and simplify the procedure of modifying objects and properties. "UV data" views display light data and other value distributions over a 2D UV parameter range.

LucidStudio also offers a number of analyzing tools: interactive ray path analysis, curvature analysis for surfaces, smoothness filters, and gradient analysis of light data, to name a few.

Being a Windows application, LucidStudio is primarily operated with a mouse, but its LucidShell interface makes it easy to extend functionality to adapt the applications to your special needs. You write your own tool in the LucidShell language and run it simply by clicking from within LucidStudio.

This documentation has two purposes: it introduces you to the use of LucidStudio, and it serves as a reference handbook in your daily work.

It is assumed that you know about optics and lighting and its terminology. Unique definitions of the terminology used in this handbook are given in the glossary. It is also assumed that you are acquainted with the standard Microsoft Windows® dialogs like Open File. If this does not fit you, we recommend you to first read the help texts accompanying e.g. the standard Microsoft Windows® Wordpad Editor.

To start, please take a look at the next section of this chapter on the principles of LucidStudio. It treats important aspects of the GUI, operational concepts of LucidStudio, and the special terminology that is used throughout this manual. After that, you can follow the example given in the last section of this chapter. It was created for a quick start and guides you through the common steps of creating an optical setup in LucidStudio. Once you are familiar with the basic steps, you can modularly increase the repertoire of commands you apply.

Therefore the following chapters describe all menu items of LucidStudio step by step. The commands available to you within each menu are explained and their application dialogs are addressed in detail.

1.2. Principles of LucidStudio

This chapter will guide you through the basic operational concepts of LucidStudio and give some insight in what the important structures of its GUI are. It will tell you about the concepts of experiment description, i.e. how light sources or sensors are defined. You also learn about the different views on an experiment that LucidStudio offers. The available mouse interactions and special properties of LucidStudio will also be addressed. Let's start with a glance at the terminology.

Whether you consider it a scene you are setting up or an optical system you want to design: Whenever a complete setup is approached, it is called the "Experiment". Every experiment consists of active parts and sometimes also passive parts. The active parts are visible to the simulation and are participating in the outcome of the simulation (e.g. ray-tracing), passive parts are not (→ VPT).

The active elements are called "Shapes" in LucidStudio. So what are "shapes"? A shape is a combination of geometry and material, so this object has a form and a behaviour in LucidStudio. Thus the shape is one of the fundamental concepts of LucidStudio and is discussed in detail in the following sections. Passive parts of an experiment do not influence the outcome of it directly. They can be data you stored or material properties you designed for later use or anything else. Basically, we divide everything that is part of your experiment into shapes and non-shapes.

The "Geometry" is another basic concept. Every active element you want to design must possess a geometry in order to exist. The geometry is therefore a property of any shape. It can be a 2D or 3D geometrical object like a curve or a surface or a cylinder, but also a single point is considered a valid geometry. Geometries can also be exchange data from and to external CAD software. Please note that a pure geometry without further properties will be considered as a passive element and therefore is not a shape.

The "Material" is the second property of a shape. Whether you define a light-source or a lens, for example, any valid shape must also consist of a material. For convenience, the material properties available within LucidStudio are split up into the three classes of emitters, actors and sensors.

"Sensors" are to some extend an exception to the other elements within LucidStudio. They are not only a material property but can be stand-alone entities themselves. In that case, they still possess a geometry, but it will be of infinity-type, e.g. an infinitely distant or infinitely large detection area. This is usually pre-defined for such a sensor type. In consequence, not all available sensors of LucidStudio can be attached to finite geometries.

The "Views" of LucidStudio are separate windows that show different parts or aspects of your experiment. Thus, they serve different purposes for design or analysis of your optics setup. Views can be considered the main user interfaces of LucidStudio and are dealt with in Section 1.2.2, “Views”.

Almost every object in LucidShape has four "Attributes" (or flags) : Editable (E), Visible (V), Pickable (P) and Traceable (T). An exemplary TreeView is shown in the picture below (see Figure 1.1, “The Model Tree View (with marked EVPT flags)”). All four flags can be toggled pressing Ctrl+Shift+Letter, just mark a shape, highlight either the Geo- or the TreeView and press the respective hot key.

"Editable" is kind of a protection against accidental changes. If unchecked, the respective shape can be examined (to see the options and parameters), but it cannot be altered. To change the values just enable the flag "E" and open the edit dialog as usual.

"Visible" determines whether this object is visible in the geometry view or not (used to enhance the clarity in more complex models). If "Pickable" is checked the respective object is pick-able in the geometry view, this can be set off to make it easier to mark and grab single objects. The fourth attribute is "Traceable". If checked this object takes part in the light simulation, if unchecked it is not traced in the ray-trace process (kind of 'invisible' for the rays, not to be mismatched with 'V' in the geometry view!). Every single attribute can be set independently to the other three, if an object is neither visible nor pick-able it still takes part in the light simulation as long as it is traceable.

1.2.1. Shapes

Experiments in LucidShape consist of "Shapes". A "Shape" is an object that takes part in the lighting experiment. It may be one of the following types:

- Light emitting devices, called "Emitting Shapes" or "Lightsources" in LucidShape's terminology.
- Objects that change incoming light by refracting, reflecting or absorbing it. These objects are called "Actor Shapes" or simply "Actors".
- Light measuring devices, called "Sensor Shapes" or simply "Sensors".

Any shape consist of two properties: Geometry and Material. To create a shape, you have to add one or more material properties to an already existing or newly created geometry.

Geometry

In LucidShape, a geometry object defines a certain region in space. It will be displayed in the geometry view, but unless it is part of a shape it will be invisible to the simulation. You can choose from a variety of pre-defined geometrical objects or create any object you need from scratch.

Material

Emitters

A shape of light source type is created, if an emitter material property is attached to a geometry. For modelling voluminous light sources like gas discharge lamps, simply define an enclosing surface with an emitter property.

Actors

An actor is also a material property attached to a geometric surface. You can define an actor to be an absorber, a reflector or a refractor. Reflectors can be ideal (i.e. 100% reflectance) or of more realistic quality. They can be perfectly specular, purely diffuse or may provide any ratio of scattering in between. A refractor may either be an ideal refractor or a Fresnel refractor (where a part of the incoming light will be reflected). To model a refractive optical medium that occupies a certain volume in space, define its outer surfaces with the corresponding refractive indices. Transmission losses, however, will not be modelled. They may be accounted for by using a scale factor, either with the light source, the refractive material or the sensor. You may also apply an absorption coefficient to the refractive material to model transmission losses. You will find detailed descriptions on the creation of available actor material properties in Chapter 7, *Emitter, Actor and Sensor Materials*.

Sensors

There are several types of sensors available. The main ones are: luminous intensity (candela), illumination (lux) or light flux (lumen) sensors. Sensors consist of a number of equally sized cells distributed across the supporting surface. The resolution of the sensor can be chosen by increasing or decreasing the number of cells. Also certain data files can be treated as a sensor.

1.2.2. Views

The experimental setup can be approached in six different ways:

- the section called “Model Tree View”
- the section called “Geometry View”
- the section called “Light Screen View”
- the section called “ UV Data View”
- the section called “ Script and Text View”
- the section called “Message View”

Model Tree View

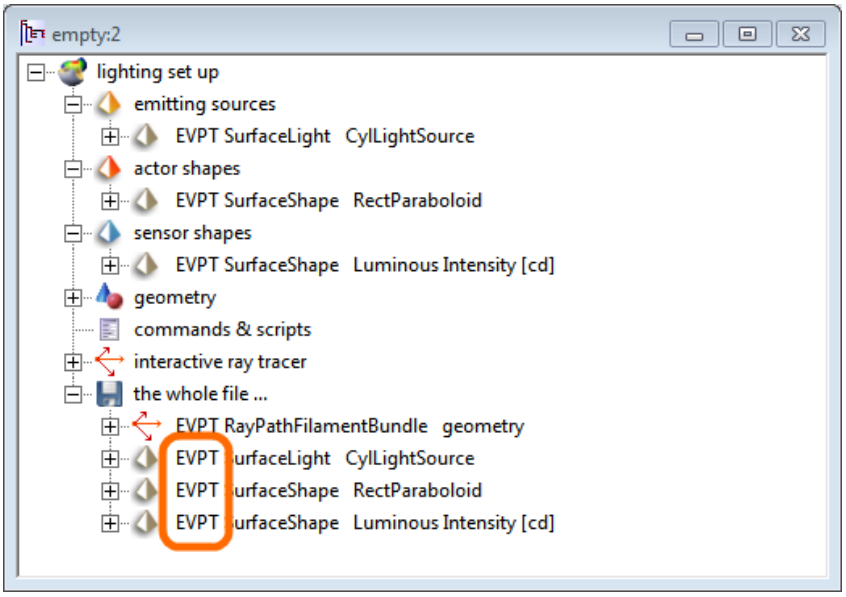


Figure 1.1: The Model Tree View (with marked EVPT flags)

In the model tree view, or simply the "tree", a textual representation of all objects in the setup and all their hierarchical dependencies are displayed. The Figure 1.1, “The Model Tree View (with marked EVPT flags)” above shows an example of the compact tree view. Once you select an entry, the object is highlighted also in the geometry view. For example, a simple cylindrical light-source (shape) enlisted in the tree would be linked to two further entries: A CylinderSurface (geometry) and a LambertEmitter (material). Even in complex experiments, you will be able to quickly identify and select one or several elements within the tree. If dialog boxes are implemented in the setup, they can be started by double mouse click directly from the tree.

Geometry View

The geometry view displays the geometric parts of your experimental setup in 3D (switchable to 2D planes), e.g. surfaces of reflectors, refractors, light sources, and detector geometries. It is also possible to display certain kinds of data associated with a surface, e.g. surface curvature or illumination data of a sensor. Also in this window, the geometry can be analyzed by using interactive ray trace.

For good orientation, an axis system is visualized. Furthermore, you can zoom objects and change between different display types like light data, shaded, black & white, and wire frame. Objects can be selected by double clicking (left) with your mouse. You can for example also choose between selecting geometries only or the complete shape (see. Section 1.2.3, “ Mouse Interaction”).

A good example of the geometry view is shown in Figure 1.2, “The Geo View”. The experiment on display consists of two lenses, one of them surrounded by a shield, a light-source and a parabolic reflector. A ray bundle has been drawn to visualize the imaging and the vertex of the reflector has obviously been placed in the center of the coordinate system. The display mode was chosen "shaded".

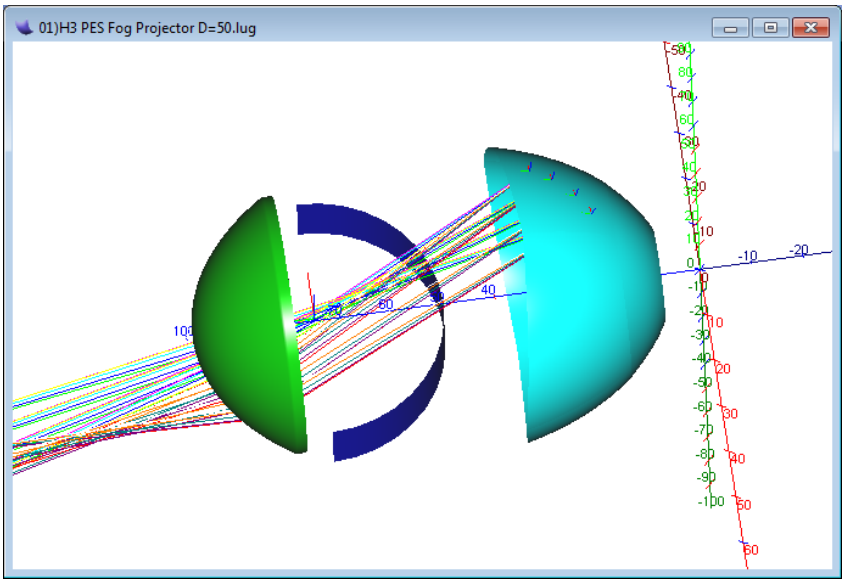


Figure 1.2: The Geo View

Light Screen View

The light screen displays filament images or ray target dots during the interactive ray trace analysis (see. the section called “ Interactive Ray Trace”). An angular coordinate system A or B, or on the plane Z at a specific position can be chosen. It is a very useful tool to ensure correct imaging and image orientation. It also can assist e.g. in estimating the impact of alterations to your optical system before doing a complete simulation.

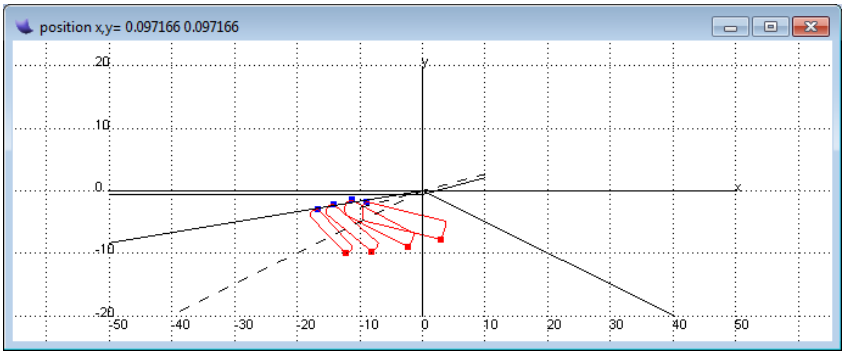


Figure 1.3: The Light Screen View

The light screen shown in Figure 1.3, “The Light Screen View” features cylindrical filament images from four different ray bundles (i.e. four separate images). During the interactive ray path analysis, any change in the optical setup directly affects these images according to the ray bundles you had previously defined. Therefore you can immediately see the impact of your modifications. Please note that the images are directly obtained from interactive ray tracing and do show different sizes correctly as a result of the actual imaging system and the path of the traced ray bundles.

UV Data View

The UV data view offers the best support for viewing scalar data like light intensities on a surface. The data are displayed according to local coordinates ("u,v"-coordinates) on the surface and they are color coded. Several analysis methods for the data are available, e.g. iso lines, smoothing (filtering), linear or logarithmic scaling.

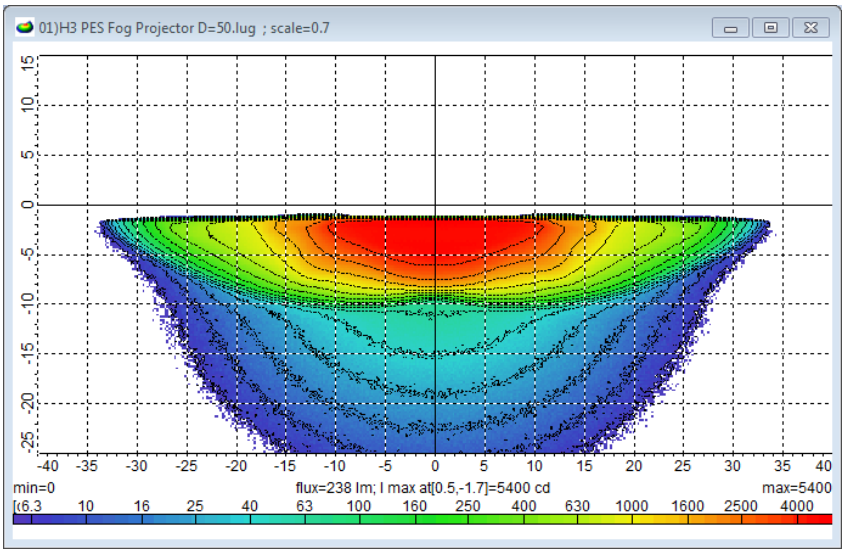


Figure 1.4: The UV Data View

Script and Text View

The text view shows a textual representation of the experimental setup as it is contained in the file. It provides you with all the information on the experiment, all the implemented shapes, and ray-tracing data as well. In the text view, complete control of the objects' parameters is possible. It is therefore recommended for experienced users of LucidShape and LucidShell. The script view is a textual representation as well, but it is used for script files with file name extension "*.do" or "*.hdo". The main feature of a script view is the "execute" functionality.

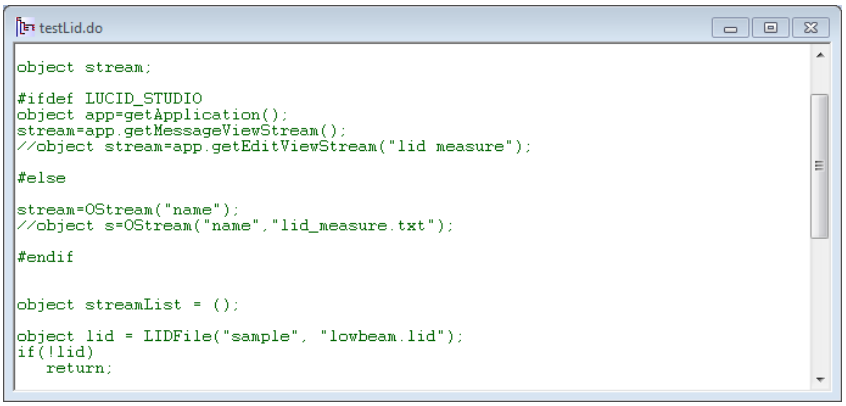


Figure 1.5: The Text View

Message View

The message view is a text view used for displaying log messages. It is strongly recommended to keep the message window always open in your work space. Errors can most easily be located by examining the log messages. All log output is also stored in a file called "LucidShape.log".

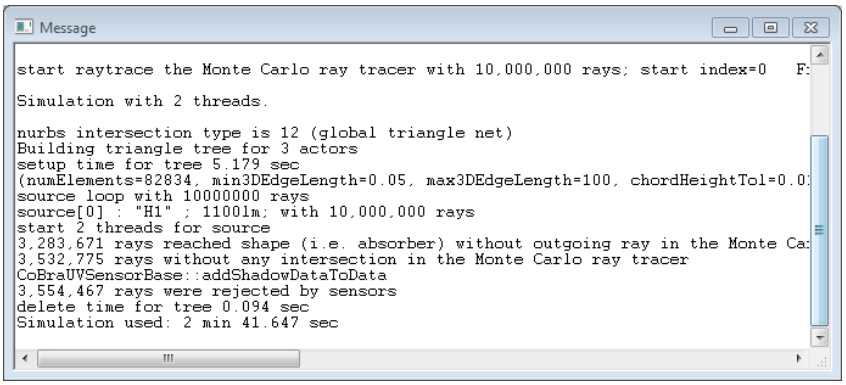

Message
start raytrace the Monte Carlo ray tracer with 10,000,000 rays; start index=0
Simulation with 2 threads.
nurbs intersection type is 12 (global triangle net)
Building triangle tree for 3 actors
setup time for tree 5.179 sec
source loop with 10000000 rays
source[0] : "H1" ; 1100lm; with 10,000,000 rays
start 2 threads for source
3,283,671 rays reached shape (i.e. absorber) without outgoing ray in the Monte Ca
3,532,775 rays without any intersection in the Monte Carlo ray tracer
CoBraUVSensorBase::addShadowDataToData
3,554,467 rays were rejected by sensors
delete time for tree 0.094 sec
Simulation used: 2 min 41.647 sec

Figure 1.6: The Message View

Beside error and warning messages, there are four common messages concerning simulation results:

- ... rays stopped because of a trace level › 'recursion'

  This means that the stated number of rays have been stopped because they have reached their limit of intersections (recursion).

- ... rays stopped because of an intensity ‹ 'intensity level'

  A ray will be considered as absorbed (and stopped tracing) if its 'intensity level' is smaller than a given critical value (mainly due to absorption processes).

- ... rays reached shape (i.e. absorber) without outgoing ray

  A ray has reached an absorber and will not traced any further.

- ... rays without any intersection in the Monte Carlo ray tracer

  This states the number of rays which had no intersection at all during the ray trace, these are the so-called direct rays.

- ... rays were rejected by sensors

  Here a ray is rejected by a sensor, i.e. it is not counted. This is because its current refractive index differs from the the settings in the sensor. Per default a sensor only detects rays with a refractive index of 1.0 (air), but of course it can be adjusted to any other medium (e.g. inside a light-guide).

- Tessellation Warning: Ran out of auxiliary vertices!

  This message may occur in former versions of LucidShape during the tessellation of complicated surfaces, when the number of triangles or vertices needed to tessellate the surface correctly exceeded a certain threshold. It is an error message, which means that the related surfaces cannot be simulated via the triangle method. This of course has a negative effect onto the simulation result.

  Newer versions of LucidShape have a better algorithm to control the number of triangles, which in most cases avoids this error. To make sure that the new algorithm is used, open the dialog via Menu → Options → Global Settings → Graphics. Change the value for *maximal auxiliary nodes per surface* to 0. Zero means that as much triangles or vertices as necessary are created to fulfil the *surface tessellation parameters 1*.

1.2.3. Mouse Interaction

In the following 'LMB' stands for 'Left Mouse Button' and 'RMB' for 'Right Mouse Button'.

RMB	In all views, pressing the RMB opens the context menu
LMB (Geo View)	• There are tree different modes within the Geo view: • G) Normal or Select Geometry mode (G BUTTON), S) Select Shape mode (S BUTTON), P) Select Pivot Point mode (P BUTTON), ESC) Toggle mode (ESC BUTTON). • Different mode enable different cursor symbols. • G) To enter the geometry or normal mode, press the button "G". Select a geometry by double-clicking the LMB. • S) To enter the select shape mode, press the button "S". Select a shape by double-clicking the LMB. • P) To enter the select pivot point mode, press the button "P". Double-clicking the LMB to select a pivot point. A ray will be shot into the scene at the position where you double-click with the LMB; this position will become the pivot point. Subsequently, the geometry will be shifted in a way that the Pivot point is at the center of the window.

	• ESC) By pressing the ESC button, one can toggle between the different modes.
LMB: Select (Tree View)	If you select an entry in the tree view, the object will be highlighted in the geometry view.
LMB: Select (Data View)	In data view, after pressing the LMB, you enter the value display mode. Point to a position or move the mouse while keeping pressed the LMB. The value of the current position is displayed in the window's entry bar.
LMB: Select (Light Screen View)	In light screen view, after pressing the LMB, you enter the position display mode. Point to a position or move the mouse while keeping pressed the LMB. The current position is displayed in the window's entry bar.
Double-Click on LMB: Select (Geometry View)	Selecting an entry in the geometry view. Depending on the modus (G,S or P) a geometry, a shape or a Pivot point. The selected geometry or shape is highlighted.
‹SHIFT› plus Double-Click on LMB: Another Select (Geometry View)	Selecting another entry in the geometry view. Depending on the modus (G,S or P) a geometry, a shape or a Pivot point. The selected objects are highlighted.
‹CTRL› plus LMB: Another Select (Tree View)	Selecting another entry in the tree view. The selected objects are highlighted.
‹SHIFT› plus LMB: Select Region (Tree View)	When you select two entries in the tree view, both entries and all entries between them are highlighted.
LMB: Rotate (Geometry View)	In the geometry view, after pressing the LMB, the rotation mode is started: Move the mouse while keeping pressed the LMB to rotate the scene.
‹SHIFT› plus LMB: Zoom Model (Geometry & Data & Light Screen View)	This combination starts zoom mode in geometry, data and light screen view. • *Zooming in:* moving the mouse upwards while keeping pressed ‹SHIFT› and LMB. • *Zooming out:* moving the mouse down while keeping pressed ‹SHIFT› and LMB.
‹SHIFT› and ‹CTRL› plus LMB: Zoom Rectangle (Geometry & Data & Light Screen View)	To draw a zooming rectangle, press both ‹SHIFT› and ‹CTRL› together with the LMB. Releasing zooms with the rectangle as view. Available in geometry, data and light screen views.
‹CTRL› plus LMB: Interactive Ray Trace (Geometry View)	This combination defines an interactive ray trace: A light ray is emitted from given start point(s). The initial direction of the light ray is given by the line(s) from the start point(s) to the point of the surface the mouse points to. If the mouse points into a geometry-free region, no light ray will be emitted.
‹CTRL› plus Double-Click on LMB: Interactive Ray Trace (Geometry View)	Freezes the current traced ray path.

Middle (Wheel) Mouse Button or Left and RMB: Panning (Geometry & Data View)	In geometry or data view, you can pan (move left/right and up/down) the scene by using the middle mouse button (or the left and right button simultaneously). In both views, this is a two-dimensional movement on a plane parallel to the viewer.
‹SHIFT› and RMB (Geometry View): Display Coordinate System	Pressing the RMB in the geometry view while holding ‹SHIFT› pressed displays a little coordinate system at the point of the surface the mouse points at. Two axis of the coordinate system visualize the tangents of the two coordinate lines (red=u, green=v). The third, blue, is the normal.
‹CTRL› plus Middle (Wheel) Button or Left and RMB	After pressing ‹CTRL› and the middle (wheel) button (or left and right button simultaneously), the ray trace start position of bundle rays is moved. This only works if rays have already been frozen, see: Interactive Ray Trace (Geometry View).
Mouse Wheel (Geometry and Data View): Zoom In and Out	Within the geometry and the data view, one can zoom in and out by moving the mouse wheel in upper (zoom in) or lower (zoom out) direction.

1.2.4. Peculiarities of LucidStudio

Dialog Box Techniques

Most dialogs offer you to assign something to an existing object or to choose one or several objects or external data files for the intended operation. These dialogs always follow the same sequence.

In dialogs with "sel" buttons (object selection buttons), first click the button. A new dialog box with "sel done" appears. When you have selected the object you desire, press the "sel done" button and the dialog will close. This is the standard procedure for this type of dialog selection.

For dialogs with "..." for file selection, files are selected by pressing the file selection button "...". A file selection box appears where you enter a file name or select the required directory. When you have finished selecting the file or required directory, press the "Open" button.

Color selection works in the same way, but the dialog selection is compelled with the "OK" button.

Script integration

One of the basic concepts of LucidShape is that scripts can be easily integrated. If you need to extend LucidShape's functionality, simply write a script to do the task. There is also a script interface to interactive dialogs, and you can integrate your own script functions in the menu tree of LucidStudio. The task "Test Scene" (from the Tests menu) is an example for an interactive script function.

Interactive Ray Trace

An important and powerful feature is the interactive ray trace. It allows you to check the optics of your experiment in an instant. The displayed ray traces are created by so called "ray trace bundle" objects. The bundle starts at the current ray start position (set in "Properties" dialog box) and goes over the current "touched" surface. The default is one single ray only. For different purposes, you can have different types of bundles, i.e. a filament bundle or a parallel bundle. You may create them with the dialog from the default task file. Also you may edit them in a separate file and add the file to your geo-view with "Add file" or "Add recent file".

You can ask for a light screen view. This view displays the images (filaments or set of dots) of the ray path bundles on a 10m screen.

1.3. The First Example: A Guide to LucidStudio

In this section, we will guide you through a simple example of the design of an optical system in LucidStudio. Keep in mind, that we need at least three different shapes: one light-source, one actor and one sensor. We will build a lighting experiment that consists of:

- one cylinder light source,
- one parabolic reflector,
- and two sensors.

There are two ways to create this example system. You can start a semi-automatic sequence of what has to be done to set up the experiment - or you can follow the instructions in this chapter and do all the necessary steps manually. You can run the automated example via the main menu: Tests → Test Scene.

After that, you may always hit the return/enter button or press the apply button in the dialog with every new dialog that appears, and you will have finally created the scene. The window that appears after the first dialog shows the geometry where you see the model in front view. To change the size press SHIFT and the LMB simultaneously in upper (increase) or lower (decrease) direction. You may rotate the model by pressing the LMB and simultaneously move the mouse inside this window. This view is called the "Geometry View". The cylinder light source is drawn in yellow or the color you have selected. To move the model inside the window press the left and the right buttons simultaneously and move the mouse. After all dialogs have been applied, you can proceed with a simulation.

It is more educational to select all menu items manually and create the example yourself. We will guide you through this process step by step. You will be able to understand and use some of the necessary procedures and dialogs intuitively. If so, you may quickly go through the steps you already can handle and proceed. Other tasks may be more complex and you will not be familiar with these. However, this chapter focuses on the creation of the example above. It will not teach you all the details and the variety of options in LucidStudio. But it presents you a task where you can follow a fundamental design sequence that is considered representative for the principles of use of the LucidStudio software.

1.3.1. Create an Empty Geometry File

To start with, we first start LucidShape and create an empty geometry file.

1. Choose "File → New" from the menu and then select "geometry" from the emerged Document Type dialog. A blank window with a coordinate system appears, it should be similar to this:

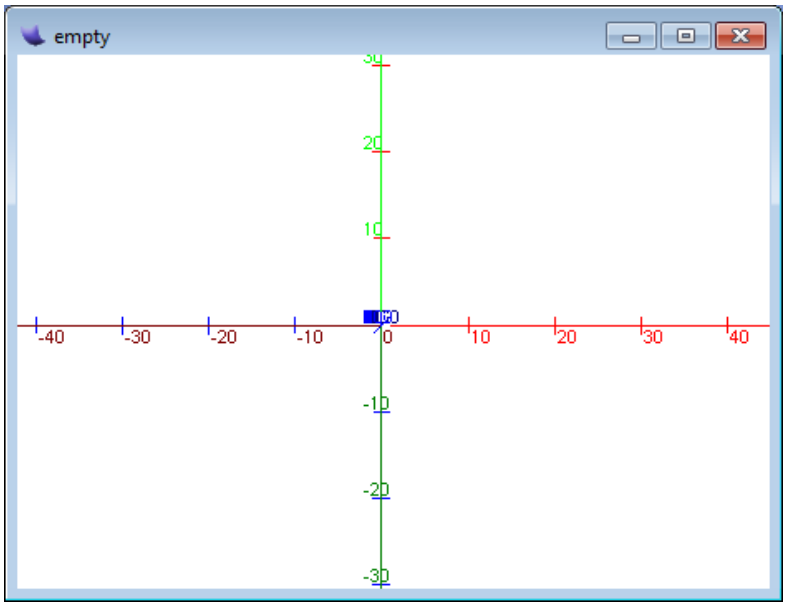


Figure 1.7: Empty Geometry View

1.3.2. Create a Cylinder Light Source

Next we insert a cylinder light source.

1. From the menu, select Lightsources → Cylinder Light Source. A new dialog window opens on your screen, in this dialog we customize the below listed options and parameters.

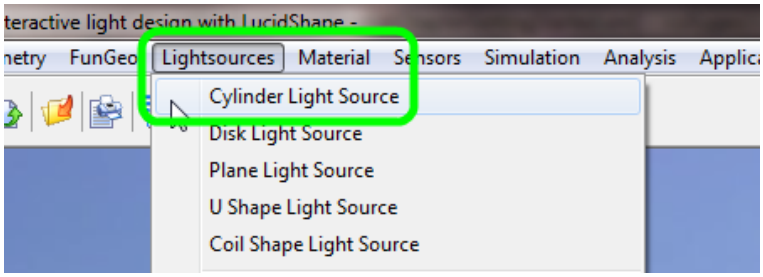


Figure 1.8: Menu → Lightsources → Cylinder Light Source

2. Let us choose the source position to be (x,y,z) = (0,0,30) and make sure, the z-axis of the cylinder is set to (0,0,1). This aligns the rotational axis of the cylinder (its local z-axis) parallel to our global z-axis.

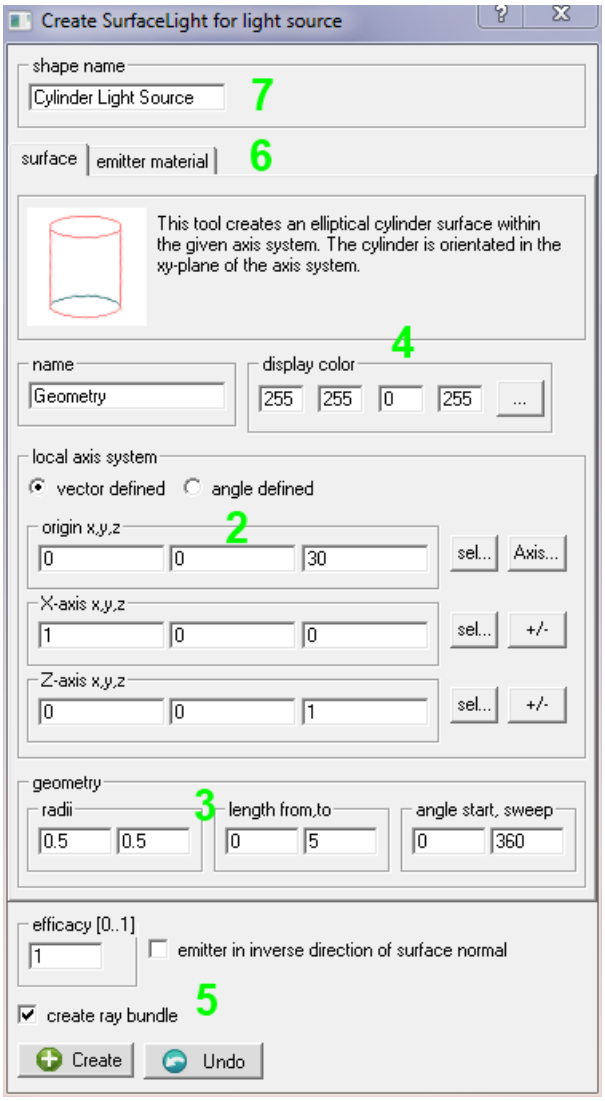


Figure 1.9: The Cylinder Light Source Dialog with the customized Options and Parameters

3. Set the length to 5 and the radii to 0.5.

4. The default display color for light-sources is yellow (255,255,0), of course it can be modified via changing the three numbers of the RGB code or select the color table via "...".

5. Check "create ray bundle" at the bottom with the ray bundle for interactive ray-trace dialog to be able to do interactive ray trace later. Alternatively, with "grid on touch surface" you could create a u by v grid where the rays touch the surface. But for now, we will use the ray bundle only.

6. Now switch to the right tab of this dialog, as emitter type we choose a lambertian emitter and set the flux to 1000 lm.

7. Enter a source name, let us take "Cylinder Light Source" and click on ‹Create›.

After pressing Create the geometry view will automatically be updated. It now contains a cylinder positioned at x=0, y=0, z=30. We see the model in front view, so it should look like a circle. Please note, what you see in the geometry view is its geometrical part, i.e. its surface. The light source is

a shape and it is represented by a surface plus the attached emitter property. To check that, take a look at the tree view. You can open it via the Menu → Window → Tree View or the TreeView Icon from the tool-bar: .

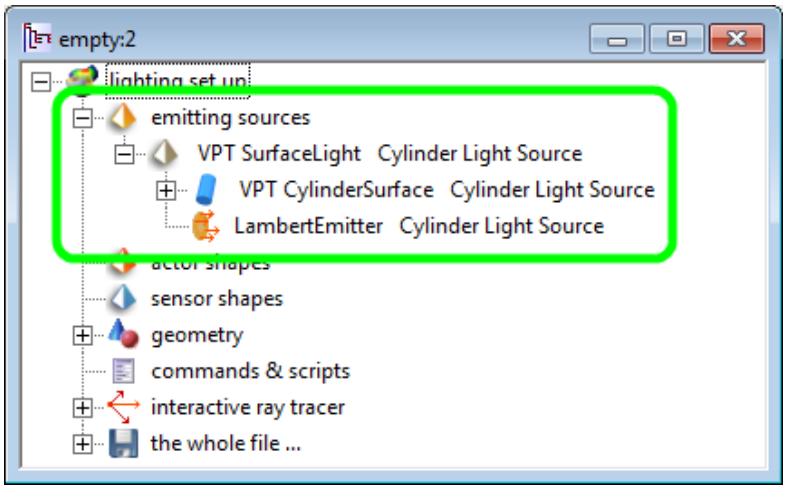


Figure 1.10: The Tree View with newly added Light-source

Our light-source appears as an emitting shape, denoted "VPT SurfaceLight". Following the tree hierarchy we also find the "VPT CylinderSurface" and "LambertEmitter" properties.

You may rotate the model by pressing the LMB and simultaneously move the mouse inside the 3D geometry window. This view is called Geo view. The cylinder light source is drawn in the specified color (if not changed yellow).

To change the size and zoom in or out, press SHIFT and the LMB simultaneously in upper (increase) or lower (decrease) direction.

To move the model inside the 3D geometry window without rotating, press the LMB and RMB simultaneously and move the mouse around while keeping pressed.

1.3.3. Define a Reflector Surface

Now we want to define the reflector surface, a rectangular paraboloid.

1. From the menu, select Geometry → Create Surface Shape → Rectangular Paraboloid.

   A dialog box opens with several sections to specify the details of your paraboloid in analogy to the previous light-source dialog box.

2. The top section defines the local axis system. Let us put the paraboloid at the origin (x=y=z=0).

3. The next section defines the geometry. Set both x_{min} and y_{min} to -50, both x_{max} and y_{max} to 50, thus covering the region [x]=[y]=[-50,50]. Set the focal length both in x- and y-direction to 30. Select a color and a name for the reflector. This defines the geometry.

Next we want to assign a property to this surface geometry: We want the paraboloid to reflect incident light, in LucidShape's terminology, we want to attach an "actor" (material) to it. This will create an actor shape.

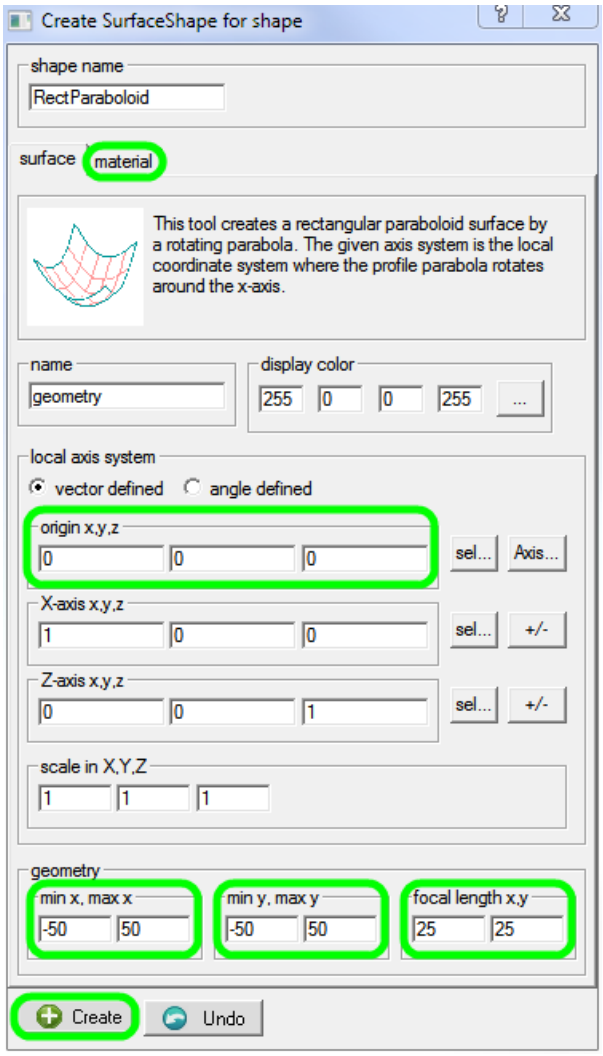


Figure 1.11: The main Dialog for Geometry

4. To do this we switch to the second tab "material" in this dialog. Within we scroll down menu we choose "Ideal Specular". Furthermore we keep the default value for reflectance, the value 1.0 means perfect or ideal reflection (= 0% losses). We apply this by pressing "Create".

Now our geometry consists of a light source and an ideal reflector.

1.3.4. Create Sensors

Next we want to see what lighting effect our device has, thus we define a first and a second sensor.

1. To create a plane geometry, choose the menu Geometry → Standard Geometry → Plane. The dialog window for plane geometries opens up, to some extent similar to the one for the paraboloid. We want the plane to be parallel to the (x,y)-plane of our global axes system and located at z=50, to do this enter these values to the respective parameters:

   Origin (x,y,z) = (0,0,50); z-axis (x,y,z) = (0,0,1); x-axis (x,y,z) = (1,0,0); x_{min}=y_{min}=-50; x_{max}=y_{max}=50. Thus, the global z-axis is the also the direction of the normal vector of the generated plane.

2. As we want this plane to be a sensor, we press "Create Shape", switch to the sensor tab and choose "Sensor for Illumination [lx]" in the scroll down menu. In the detector setting, we set the cell size to 1 and keep the rest of the default settings. Press "Create" to create our lux sensor.

3. Next we create a luminous intensity distribution sensor. Select from the menu Sensors → Candela Sensor. Use the default values and confirm them by pressing "Create", a candela sensor is created.

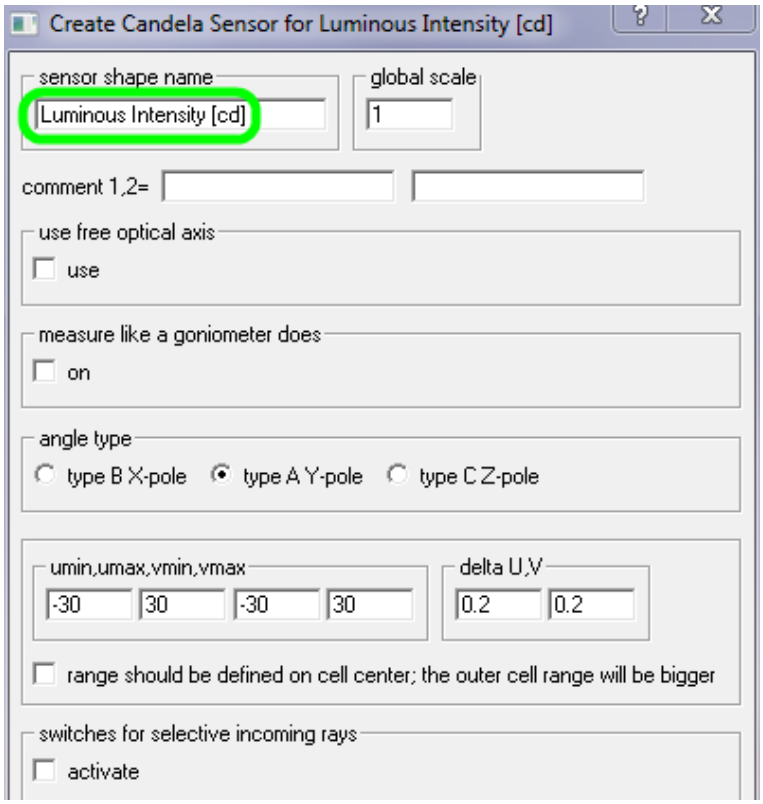


Figure 1.12: The Candela Sensor Dialog

Please note, this [cd] sensor differs from the [lx] sensor: it is neither attached to the plane you just created, nor any other of your geometries. The candela sensor requires an infinity surface geometry (check tree view!). Thus, it is not allowed to assign the candela sensor to an arbitrary finite surface.

⇒ Now our experimental setup is complete, we have an emitter, an actor and at least one sensor. The GeoView and the TreeView respectively should now look similar to this:

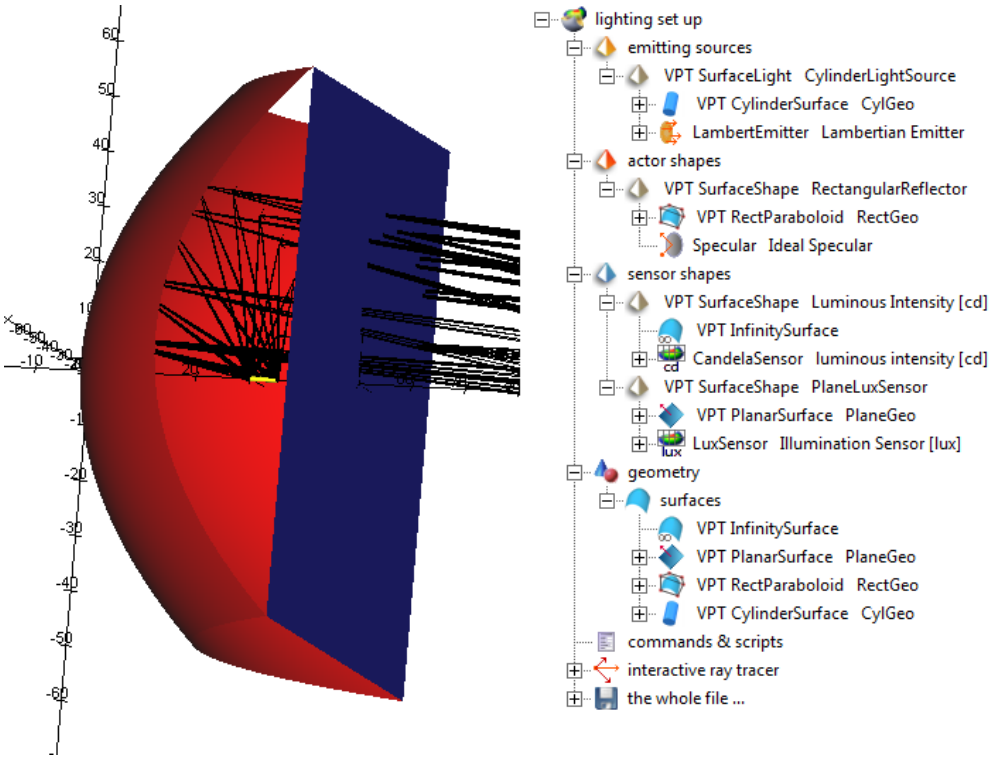


Figure 1.13: The Model with Light-source, Reflector and Sensor (left GeoView, right TreeView)

1.3.5. Simulate the scene

Now we want to simulate the scene by the Monte Carlo ray trace algorithm in order to get some results on how our optical system performs.

1. Select Menu → Simulation → Simulate by Monte Carlo.

2. The ray trace dialog has appeared. As number of rays we enter 2000 (internal multiplied with *1000), but keep the default values for the other parameters and press "Start" to start the simulation. If available on your system, make sure that "Multiprocessing" is activated.

   ⇒ After the simulation has finished (watch the progress bar in the status line at the bottom of the window), two light data views are visible, one for each sensor. A lux sensor data view is also shown in Figure 1.15, “UV Data View: lux sensor (left picture) and cd sensor (right), both unfiltered”.

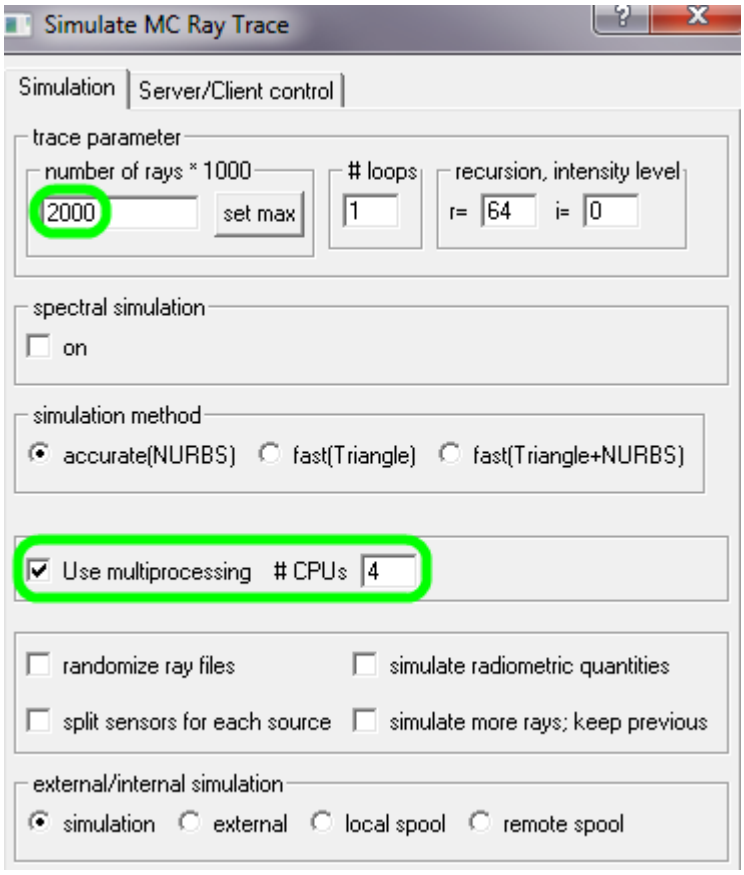


Figure 1.14: Ray Trace Dialog

3. In case you have closed a light data view, choose "All UV / Light-Data Views" either from the context menu (press RMB) or from the Menu → Window → All UV / Light-Data Views or you can press the tool-bar icon to recreate it. Now one view shows the illumination data ("lx sensor") directly in front of the reflector, and the second view contains the luminous intensity distribution in Candela ("candela sensor").

4. You may smooth the view by filtering (press one of the tool-bar icons). Filter it a few times, until the view is sufficiently smooth.

5. Alternatively, to obtain a smoother image, you may want to restart the simulation with a larger number of rays 10000 (*1000) to get 10,000,000 rays simulated.

6. You also should look at the message window (right lower part of the screen) where certain actions are logged and the total light flux of the device is given.

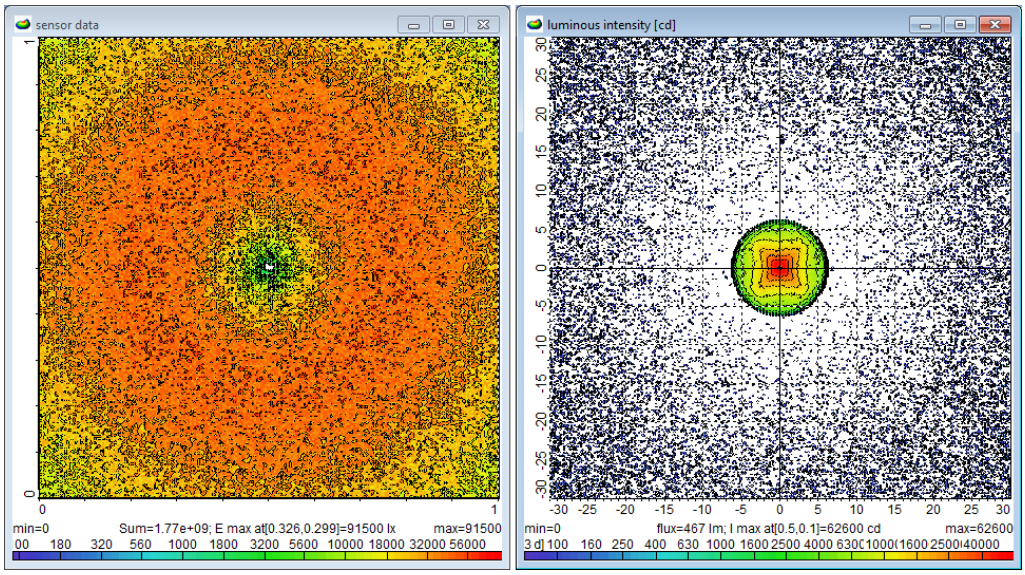


Figure 1.15: UV Data View: lux sensor (left picture) and cd sensor (right), both unfiltered

The UV Data Views provide you with the sensor data. You have two lux sensors (the plane and the paraboloid) and the candela sensor. The latter is shown in Figure 1.15, “UV Data View: lux sensor (left picture) and cd sensor (right), both unfiltered”. The context menu (RMB) of the UV Data View or the menu UVView offers you a variety of options to post process your results. You will find details on the available commands in all UV Data Views in Section 3.4, “UV View” of Chapter 3, *Views*.

7. Next we want to analyze the effect of particular parts of the reflector on its imaging properties, so we perform an interactive ray trace. Open a light screen view (while in the geometry view either select from the context menu or press the tool-bar button .

8. Press ‹Ctrl› and the LMB while pointing somewhere on the reflector. In the geometry view, you will see a few rays emanating from the light source, while in the light screen view the image of the light source on a wall is displayed.

9. Move around with the mouse while keeping pressed ‹Ctrl› and LMB simultaneously.

10. ‹CTRL› plus Double-Click on LMB in the Geometry View freezes the current traced ray path.

11. Select the data view window where the plane with the illumination sensor is displayed. Select "Properties" from the context menu, deselect logarithmic scale and press ‹OK›. Finally filter the display (one of the tool-bar buttons). The linear scale better exhibits the regions where the illumination obtains its extreme values, while the logarithmic scale better corresponds to the physiological effect (i.e. how the light is sensed).

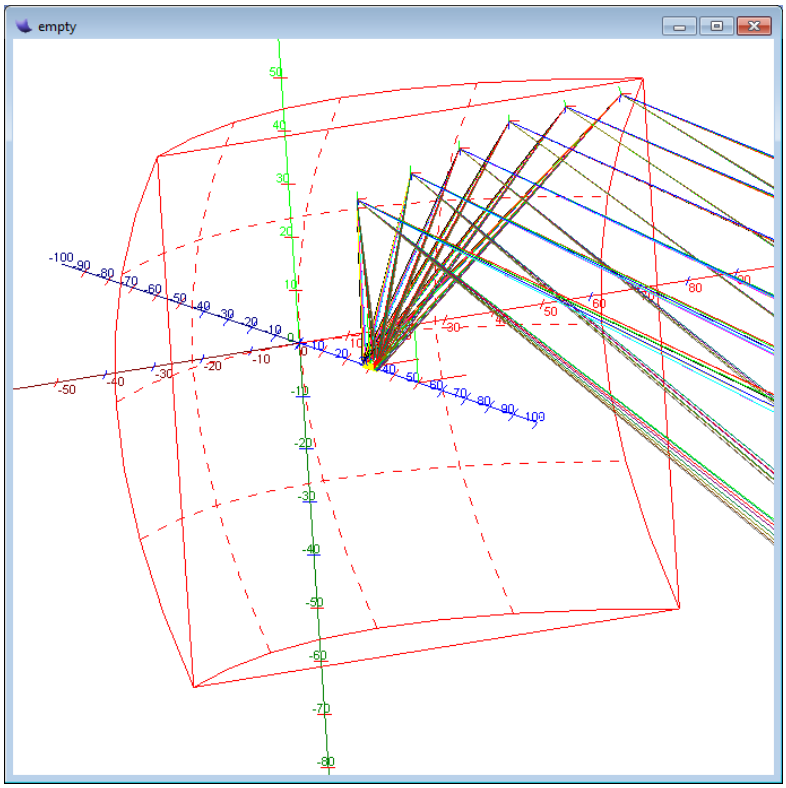


Figure 1.16: Geometry View: Interactive Ray Tracing

The complete setup of this example together with some additional informations and pictures can be found in the first of our tutorials: Menu → Help → Tutorial → First Steps.

Chapter 2. File and Edit

This chapter describes all commands in *LucidShape* that are available under the menus *File* and *Edit*.

Menu entries are referenced in the form 'Menu → Menu Item → Next Menu Item → ...'.

2.1. Editing LucidShape Documents

The following menu entries are available under *File*:

new	Creates a new empty document. Choose the appropriate file type from the menu that appears. By this, the view to be opened is defined.
open	Opens a file and displays its contents according to the file name suffix. The file to be opened can be selected in a file selection box. For a list of default LucidShape files, see Appendix A, *LucidShape File Types*.
open again	Reloads the document in focus from its associated file and updates all of the document's open views.
close	Closes the document in focus and closes all of the document's open views.
open file for text edit	Opens a file and displays its contents in a text edit view.
show in text edit	Displays an already open file in a text edit view.
add	Adds the contents of a file to the document in focus.
add recent	Like *add*, but the added file is selected from a list of previously opened files.

2.2. Saving LucidShape Documents

The following menu entries are available under *File*.

The general type of the object is determined automatically and cannot be changed (e.g. a LID cannot be saved as a .lug, this extension stands for a geometry file). With a fixed general type, the suffix can be chosen among a wide variety of different file formats, of course depending on the type of the respective object. A LID e.g. can be saved as .lid, .csv, .lmt, .ies, .cie, etc. (Due to this scanning for the correct file suffix, it should be avoided to have a dot in the file name.)

save	A previously saved object (like e.g. a complete experiment, a GeoView, a LID, etc.) can be saved into the same file as before, i.e. overwriting this file. An optional warning concerning this overwriting can be toggled on/off in the options ('Global Settings'). Please note: The default file format is binary, so to obtain an ASCII file, you have to use the 'Save As ASCII' entry.

save as	Saves the current data object (experiment setup, light data, etc) with a new file name. Please see also the note above this table and Section D.2, “Materials Import/Export: Rhino CAD”.
save selected	Saves the selected geometry with the current file format and name.
save as ASCII	Like *save*, but the document is saved in a plane text format (this requires increased disk space).
save as Metafile	Saves the UVData view in focus as a .emf file (Windows Enhanced Metafile, a vector based graphics format). Only applicable if the view is a UVData view.
save as bitmap	Makes a screen hard-copy of the view in focus as a Windows Bitmap File (.bmp). It is possible to save the screen hard-copy as other common picture types with this command by changing the suffix of the file.
save UVData as bitmap	Saves the data of the UVData view in focus as a Windows Bitmap File (.bmp) by mapping the uv-data cells directly on bitmap pixels. It is possible to save the UVData view in focus as other common picture types with this command by changing the suffix of the file.

2.3. Transferring Geometry between LucidShape and CAD tools

The following menu entries are available under *File*:

CATIA Transfer	Opens a dialog (see Figure 2.1, “The Catia Transfer Dialog”) to perform a CAD data exchange between LucidShape and CATIA™ V5. A CATIA session is started simultaneously (if not running yet). The key approach is to define a suitable local axis system (the CATPart axis system) in CATIA™, representing a transform to be applied to the geometry on its transfer between CATIA™ and LucidShape. See [1] for detailed information on how to use this tool.
SolidWorks Transfer	Opens a dialog to perform a CAD data exchange between LucidShape and SolidWorks™, simultaneously a SolidWorks session is started.
Rhino Transfer	Opens a dialog (see Figure 2.2, “The Rhino Transfer Dialog”) to perform a CAD data exchange between LucidShape and Rhino™, simultaneously a Rhino session is started (if not running yet). Note that the correct Rhino version (2, 3 or 4) has to be set under *Options* → *Global Settings* (see Section 12.2.1, “Global Settings”).

2.3.1. The CATIA Transfer Dialog

axis vectors	The axis system which defines an extra transformation to be applied when transferring geometry between LucidShape and CATIA™.
create axis in LucidShape model	Creates the transformation axis system as an object in the active LucidShape model.
pick axis from active CATPart	Switches to the CATIA object selection mode which allows the selection of an axis system in CATIA. Once selected, the axis system will be displayed in the above axis system dialog.
Custom/Picked Axis	Drop down dialog containing a list of selectable axis systems that are in the currently active CATIA document. Once selected, the axis system is displayed in the dialog's axis system view.
update axis list	Gets an updated axis system list from CATIA, see "Custom/Picked Axis".
create all axis systems	Takes all axis systems within CATIA and creates new copies within LucidShape.
transfer file format	Allows the user to select between IGES and STEP file formats for the transfer between CATIA and LucidShape.
LucidShape → CATIA	Sub-dialog to transfer the current LucidShape model to the active CATIA document. • *transfer*: Transfers the whole active LucidShape model into the active CATIA document • *transfer selected*: Transfers only selected parts of the active LucidShape model into the active CATIA document • *info*: Provides information about the LucidShape to CATIA transfer in a separate dialog (see Figure 2.1, “The Catia Transfer Dialog”)
CATIA → LucidShape	Subdivision to transfer the current CATIA document to the active LucidShape model. • *transfer*: Transfers the whole active CATIA document into the current LucidShape model • *transfer selected*: Transfers only the selected parts of the active CATIA document into the current LucidShape model • *info*: Provides information about the CATIA to LucidShape transfer in a separate dialog (see Figure 2.1, “The Catia Transfer Dialog”).

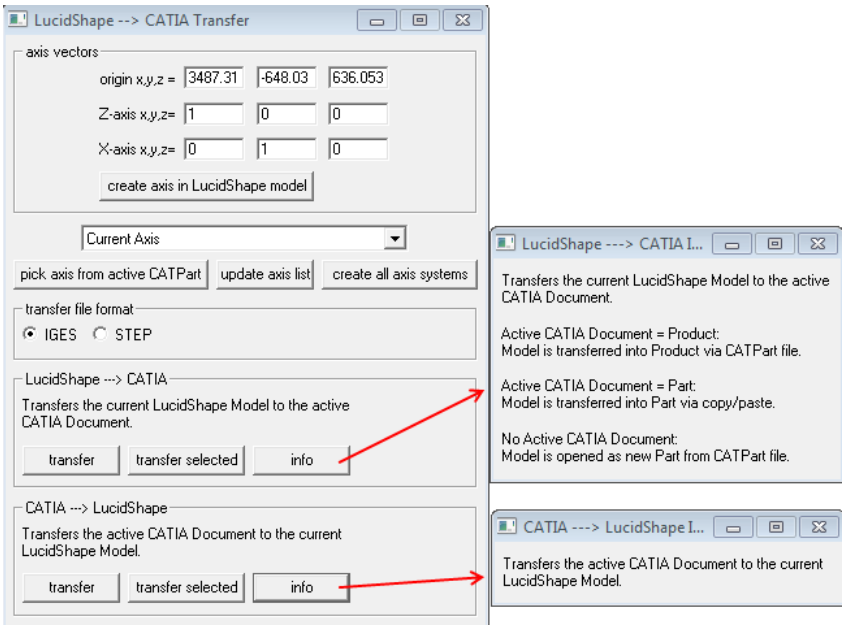


Figure 2.1: The Catia Transfer Dialog

2.3.2. The Rhino Transfer Dialog

Choose axis system for Conversion	The axis system which defines an extra transformation to be applied when transferring geometry between LucidShape and Rhino. • *pick origin:* Switches to Rhino to pick the axis system's origin point. • *pick z axis:* Switches to Rhino to pick the axis system's z axis. • *pick x axis:* Switches to Rhino to pick the axis system's x axis. • *update:* Updates the transformation axis system as an object in the active LucidShape model. • *create axis in current view:* Creates the transformation axis system as an object in the active LucidShape model. The coordinates are displayed in LucidShape after picking the data in Rhino.
LucidShape → Rhino	Sub-dialog to transfer the current LucidShape model to the active Rhino document. • *LS-Model → Rhino:* Transfers the LucidShape scene into Rhino and deletes the old existing scene within Rhino. • *Add LS-Model → Rhino:* Adds the LucidShape scene to an existing scene within Rhino. • *Selected LS-Surfaces → Rhino:* Transfers the selected surfaces into Rhino.
Rhino → LucidShape	• *Rhino Scene → new model:* Transfers the Rhino scene into LucidShape and opens a new GeoView. • *Rhino Scene → current model:* Transfers the Rhino scene into the current scene of LucidShape.

- *selected Rhino surfaces → current scene:* Transfers the selected Rhino surfaces into the current scene of LucidShape.

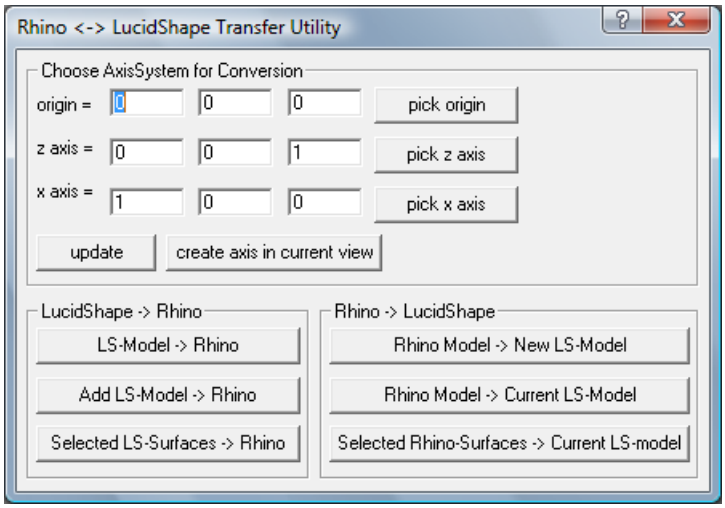


Figure 2.2: The Rhino Transfer Dialog

2.4. Transformed Input and Output

These commands allow input and output of geometry while applying a linear transformation with a company axis system (see Section 12.2.1, “Global Settings”).

open / add / save transformed	Parent menu entry for the commands described subsequently.
open transformed	Like "open", but any geometry undergoes a linear transformation by the company axis system before it is read into LucidShape.
add transformed	Like "add", but any geometry undergoes a linear transformation by the company axis system before it is read into LucidShape.
save transformed	Like "save as", but any geometry undergoes a linear transformation by the company axis system before it is saved to a file.

2.5. Menu Operations

A menu file is a .do script file containing declarations of script functions which are executable as menu items. The menu can be overwritten by your own menu script file, enabling you to extent the menu interface by programming in LucidShell language and setup the interface for your current project. For example, for a project with several calculation procedures for each reflector surface, you can make menu items for each reflector part and enter your reflection angles into your "home made" dialog box.

menu operations	Parent menu entry for the commands described subsequently.
load default menu	This entry loads the file with the menu structure which is internally predefined and hard linked to the executable file. This menu cannot be changed.
replace menu	The default menu can be replaced by an own written menu. Once selected, the file name is stored in the INI file.
add menu	Opens a file dialog to select a menu .do file, in order to add a customized menu to LucidStudio.

2.6. Printing

The following menu entries are available under *File*:

printer	Parent menu entry for the commands described subsequently.
print preview	Preview what would appear on the printer.
print	Sends the document or parts of it to the printer.
print setup	Setup printing parameters like page size and format.
print page setup	Set parameters for the printed page layout like title text or scale factor for the geometry view.

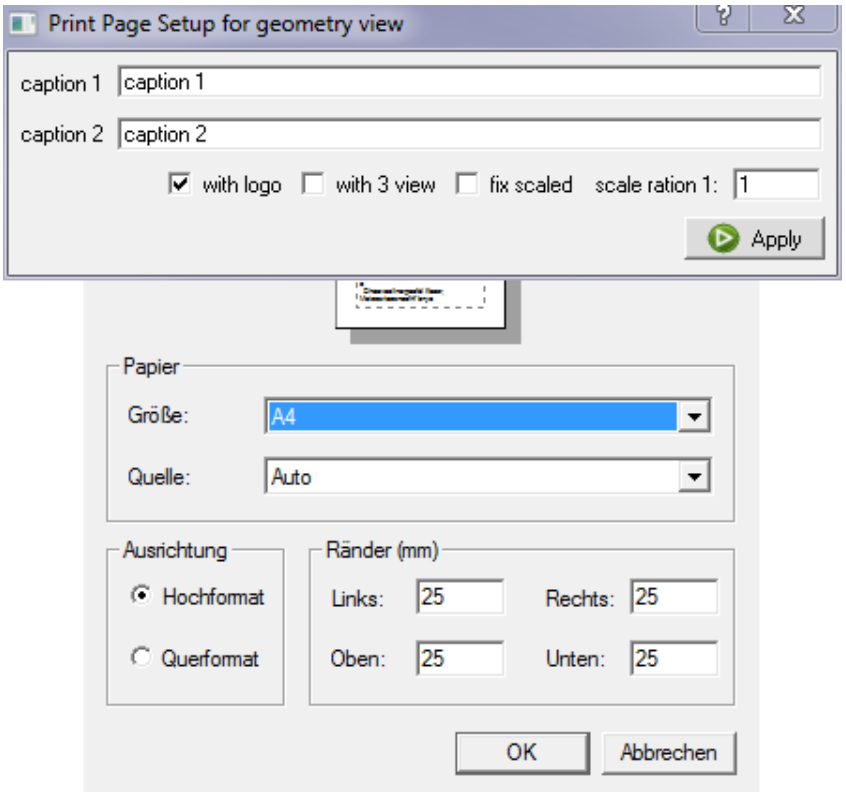


Figure 2.3: The "Print Setup" and "Print Page Setup" Dialogs

2.7. Standard Editing

The following menu entries are available under *Edit* or *UVEdit* (the latter applies for UVData views only):

undo	Reverts the last operation.
cut	Cuts the selected object (i.e. copies it to the clipboard and removes it from the document in focus).
copy	Copies the selected object to the clipboard.
paste	Pastes the contents of the clipboard to the document in focus.
copy Bitmap / Meta-file	Makes a screen hard-copy of the view in focus and copies it as a Windows Bitmap or Windows Meta-file into the clipboard, to be pasted into a graphics tool. This is only available for geo views (copy Bitmap) and UVData views (copy Bitmap and copy Meta-file).
delete	Removes the selected object from the document in focus.
clear all	Empties the clipboard.

Bibliography

[1] *LucidShape Catia V5 Geometry Transfer*.

Chapter 3. Views

Apart from the message view which usually resides in the right bottom of the workspace, LucidStudio offers you five different "Views". These are the *Geometry View, Tree View, Text/Script View, UV DataView* and *LightScreenView*. They allow different approaches to the purposes of building and evaluating an experiment, and offer you different menus with individual commands and options during the processes of design and analysis. Therefore, each of them gives you a different "View" on your project.

This chapter describes the specific commands for these five views. In LucidStudio, a corresponding menu appears for each type of view at the third position in the menu bar, and is updated dynamically, depending on the type of view in focus. So only one of the view menus is available at a time. The specific commands are available within the corresponding view menu, while a window for each type of view can be opened manually via the menu item *Window*.

- The *GeoView* shows the complete geometry of the experiment. In a variety of different graphical representations, you can freely rotate the optical system in 3D and you can select each shape for design purposes. The axis (x,y,z) are always displayed to allow a quick orientation within the setup (they can be switched off manually, see below).

- The *TreeView* presents the most compact overview on your experiment at any stage of your work. Also, you can access a dialog window for each element, either simply right-click or double left-click onto an entry in the *TreeView*. The menu offers mainly selection tools and access to properties of the tree itself or elements you selected. The listing in the TreeView is structured, every entry belongs to a main group like e.g. "light sources" or "sensors". These main folders (and their sub-folders too) can be expanded or collapsed to enhance the clarity (very similar to a file explorer).

- The *Text-/ScriptView* is the most fundamental view, yet probably the most difficult to use. The *Text-/ScriptView* allows you to directly edit the properties of all elements by editing in clear text. Main features of its menu are the update and execute commands.

- The *UVDataView* gives you access to the sensor data you have obtained from ray tracing or light mapping. The *UVDataView* menu allows the interpretation, further analysis, and post-processing of your results by a variety of commands .

- The *LightScreenView* shows you the images of your light source (i.e. filament image) on a screen, created from ray bundles you define during interactive ray tracing in the GeoView. The *LightScreenView* menu allows you to access different viewing angles and other view properties or the LightScreenView properties dialog.

The *Message View* displays all LucidStudio messages but does not have a menu of its own. All executed commands, error codes or warnings are shown in this window.

3.1. GeoView

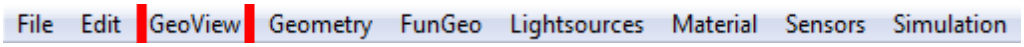


Figure 3.1: Geo View Menu

The following menu entries are available under *GeoView*:

3.1.1. General Commands

info	Shows information about the model in the message box.
re-frame	Re-frames the value range to the default range after zoom etc.
re-frame selected	Like "re-frame", but only the selected objects are considered.
redraw	Recreates and displays the scene.
auto redraw	Switches to an automatic redraw. Each time you rotate or move the scene by mouse interaction, the scene will be recreated and displayed.
zoom in/out	Zooms in/out in small steps. You can also zoom in/out by moving the mouse wheel in upper/lower direction, or by using the SHIFT and the LMB while moving the mouse up/down.

3.1.2. Selection Modes

With the commands in this menu, you can determine the object selection mode of the GeoView. The active selection refers to the treatment of objects by double mouse clicks. You can also switch via hot keys between the three modes, see Section 1.2.3, “ Mouse Interaction”.

Geometry Selection Mode	Switches to the geometry selection mode which restricts the selectable objects to pure geometric objects (surfaces, curves, ...).
Shape Selection Mode	Switches to the geometry selection mode which restricts the selectable objects to shapes.
Pivot Point Selection Mode	Switches to the geometry selection mode which restricts the selectable objects to pivot points (3D navigation reference points, must lie on a surface).

3.1.3. Viewing Perspectives

main views	Aligns the viewing axis to display the model in one of the main views (XY, YX, XZ, ZX, YZ, or ZY).
2D mode	Enables or disables the 2D view mode, which restricts panning the geometry inside a plane.
perspective/orthogonal view	Toggles between perspective and orthogonal view.

3.1.4. Clear

clear selection	Removes all selections from the current geometry.
clear ray paths	Deletes all ray paths from the geometry view.

3.1.5. Axis

stock scene	Pops up a dialog to set the display of the stock scene (the displayed main axis system). The axis system's blocks can also be scaled by powers of 10. See Figure 3.2, “The Stock Scene Dialog” and the related table.

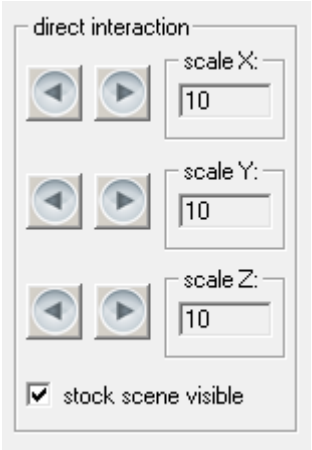


Figure 3.2: The Stock Scene Dialog

x-axis, y-axis, z-axis	These values scale the blocks of the axis system. The scale factor is 10^x.
Stock Scene Visibility	If checked, the axis system is visible, otherwise it is not.

3.1.6. Objects Coloring and Display

The first three commands in this section open a Windows color selection dialog to select the desired background color.

background color	Sets the background color of the view in focus.
3D default background color	Sets the default background color for all geometry views in 3D mode. The change will first come into effect when opening the next geometry view. It will neither affect the view currently in focus nor other already opened views unless a redraw command is executed subsequently.
2D default background color	Like *3D Default Background Color*, but for geometry views in 2D mode.
black & white	Toggles between black & white and color display.

surface display modes	Enables the desired display type. See the table in the section called “Surface Display Properties” for a description of the different types available.

3.1.7. Advanced Display Properties

The menu commands in this section open a property page to set some advanced display options.

Surface Display Properties

The Surface Display Properties option pops up a property dialog for surface display options. See the picture and the related table right below.

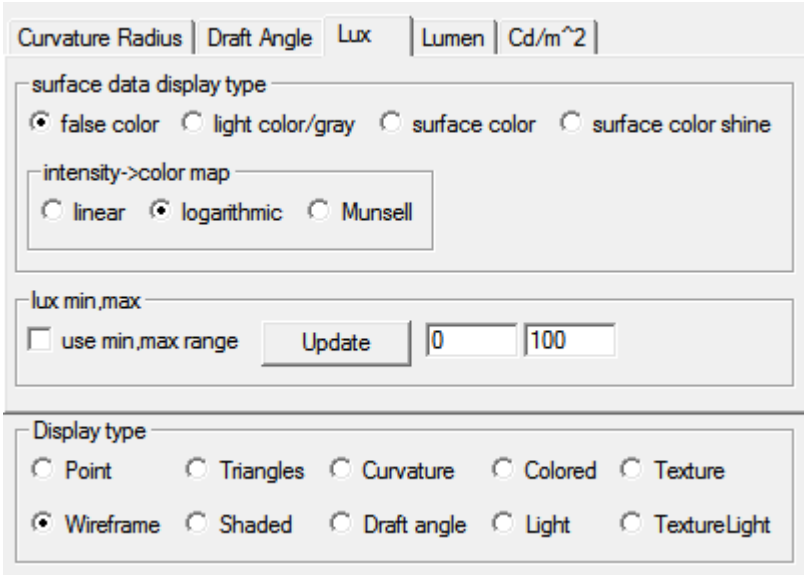


Figure 3.3: The Surface Display Properties Dialog

main data type	One can select among *Curvature Radius, Draft Angle, Lux, Lumen* and *Cd/m²* . When showing the draft angle, the draft direction can specified, per default it is set to (0,0,1).
surface data display type	The main data type can be displayed in *false color, light color/gray, surface color*, or *surface color shine*. Surface color uses the color range from black to surface color, surface color shine uses a color range from black to surface color and in addition to white, therefore, it appears shiny.
intensity → color map	Switches between linear, logarithmic and Munsell (similar to logarithmic) data display. For more information about the Munsell type, please refer also to the Technical Reference Manual.
use min, max range	If checked, the entered min-/max-range is used for the respective data type.
display type	Switches between the different display types: • *Point:* displays the geometry with a point raster.

	• *Triangles:* displays triangle meshes. • *Curvature:* the curvature of the surface is displayed in different colors. • *Colored:* the surfaces are colored, their normal directions define the color. • *Texture:* displays the texture on a surface. • *Wireframe:* displays the geometry with a wire frame. • *Shaded:* switches to shaded display. • *Draft Angle:* displays the draft angle of the geometry via a color-map on the surface. The draft angle is defined as 90°-α, where α is the angle between the surface normal and the optical axis (the z-axis). Small draft angles (surface normals nearly perpendicular to the optical axis) should be avoided. • *Light:* switches to light display mode, sensor surfaces show their light distribution. • *TextureLight:* switches to texture light mode. Texture light displays the texture and additionally the light distribution if there is light stored in the sensor. It cross fades the light and the texture.

Ray Analysis Properties

The Ray Analysis Properties option pops up a dialog to set the interactive ray trace properties. See the picture and the related table right below.

ray start position	• *ray bundle start X,Y,Z:* Defines the start point of the ray bundles for random rays and interactive ray trace. • *shift/rotate...:* The start position X,Y,Z may be shifted/rotated from its current point by the entered delta value in the external dialog box. • *depth for 2D trace:* If checked, the value defines the depth of the rays used for interactive ray trace in the 2D geometry view. • *start refraction index:* The start refraction index for the rays.
grid on touch surface	• *number of rays in u,v:* Upper bound for the number of surface intersections for a grid of rays (CTRL + RMB). • *offset to border:* Rays of the interactive ray trace will start from the surface border (edge of the surface extent in U and V) by the offset provided.
ray trace level	• *recursion:* Upper bound for the number of surface intersections for a single ray. • *intensity:* Rays with intensities below these values are ignored in the interactive ray trace.
start random rays from	Toggle the random rays starting from either ray bundles or the light source centers. The number of random rays generated each time the random rays command is triggered may also be set here.

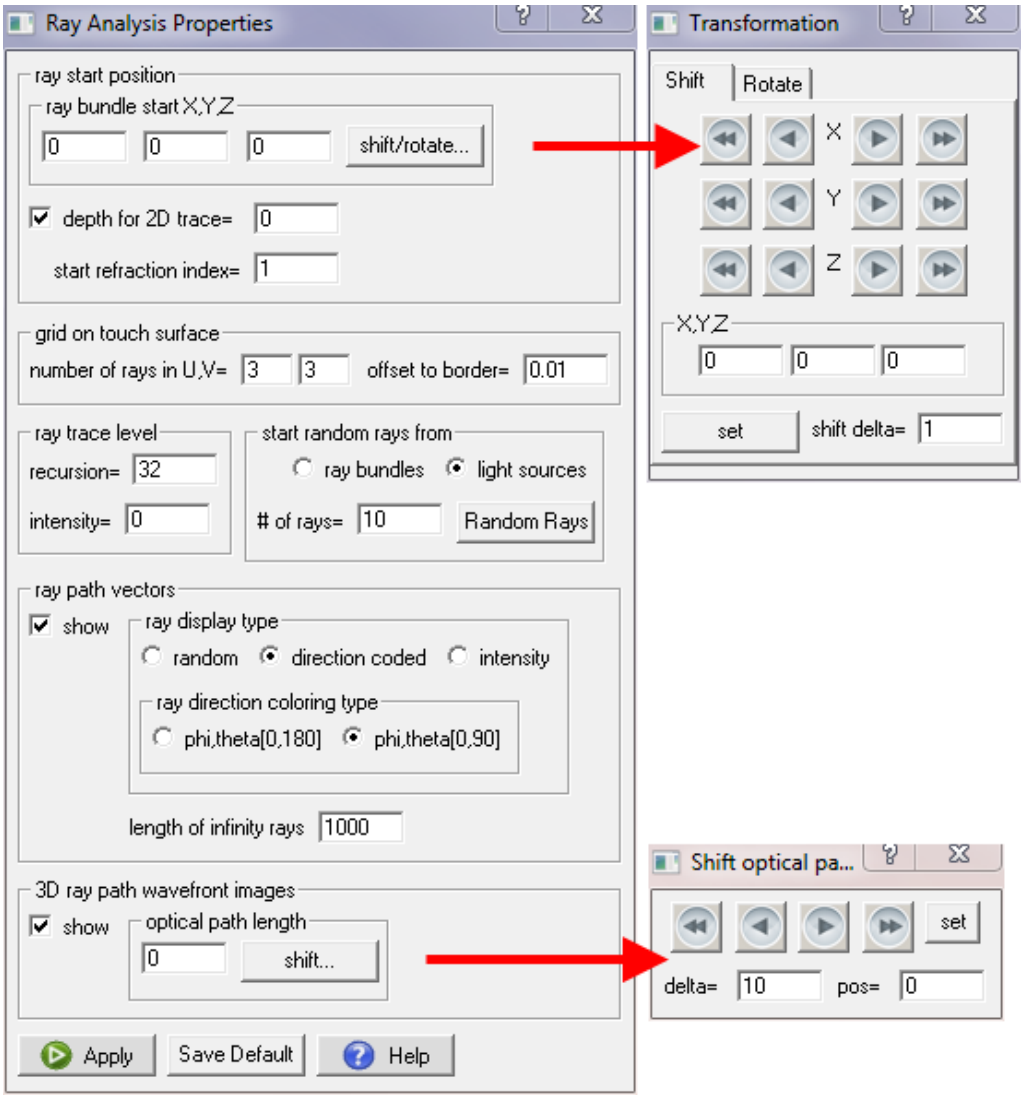


Figure 3.4: The Ray Analysis Properties Dialog

ray path vectors	Sets the generated number and coloring of rays each time the *generate random rays* command is triggered. Possible settings are: • *random:* the rays are displayed in randomized colors. • *direction coded:* The ray coloring depends continuously on the ray direction. • *intensity:* The ray coloring depends continuously on the ray intensity.
3D ray path wavefront images	Enables the display of wavefront images at a user defined optical path length. This dialog part may be switched on or off with the *show* check-box.

Camera Properties

The Camera Properties option pops up a dialog to set general viewing properties in the geometry view, i.e. viewing direction. See the picture and the related table right below.

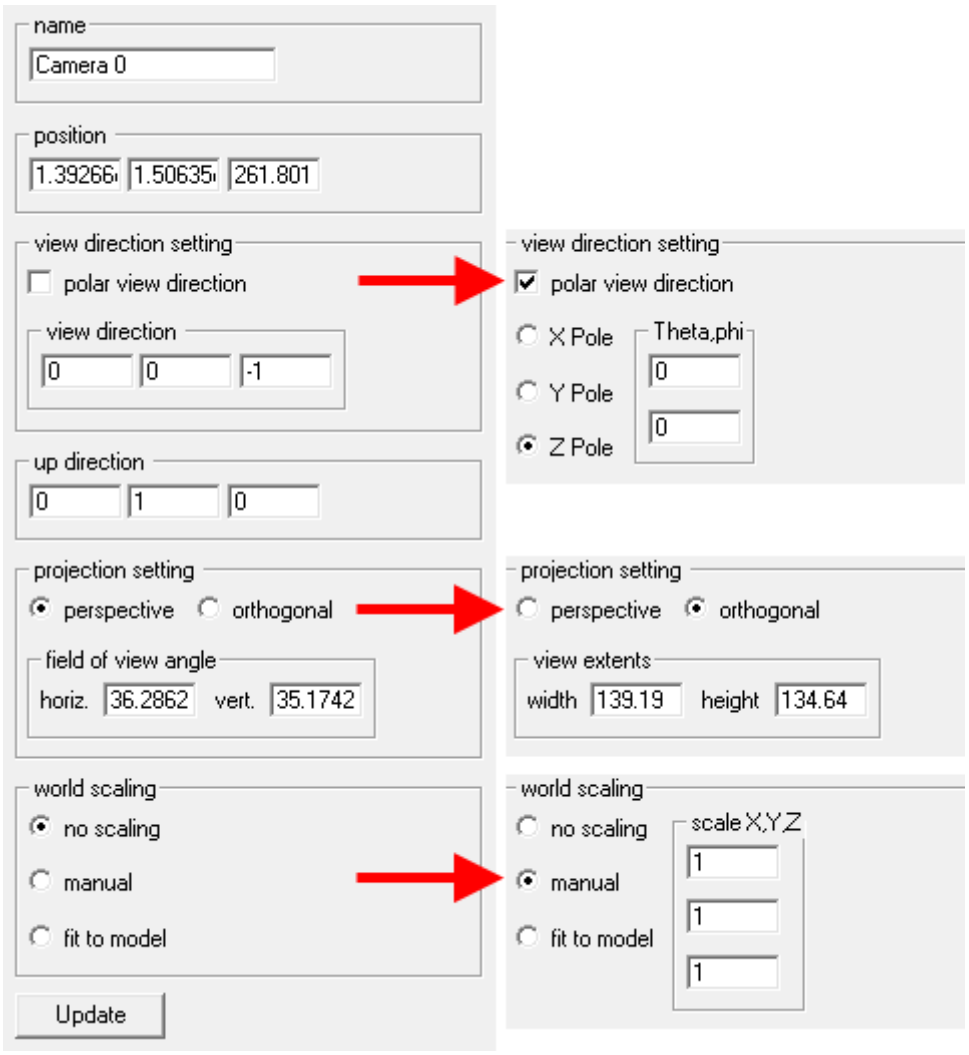


Figure 3.5: The Camera Properties Dialog

view axis	Specifies the view camera position and viewing direction. • *position:* The (x,y,z)-position from where the object or the scene is viewed. • *view direction setting:* Depending on the checked mode (polar or not polar), the view direction can be entered directly as a (x,y,z) vector, or in polar coordinates via a pole axis (x, y or z) plus two rotation angles. • *up direction:* The vector for the camera up view.

projection settings	The type of camera projection: perspective (viewing area is specified by a *field of view angle*), or orthogonal (viewing area is specified by a *view extent*). • *field of view angle:* The viewing area in the perspective case (horizontal and vertical), or: • *view extent:* The viewing area in the orthogonal case (width and height).
radio button: world scaling	This radio button allows to modify the ratio of the camera's x, y and z viewing axes, i.e. distorts the view. • *no scaling:* Does not make any changes at all. • *manual:* Changes the ratio to the specified set. • *fit to model:* Automatically adjusts the ratio to optimally fit into the GeoView, i.e. distorts the view.
button: modify	If pressed, the current settings are applied from the dialog into the GeoView.
button: update	If pressed, the current settings are applied from the GeoView into the dialog.

3.2. Tree View

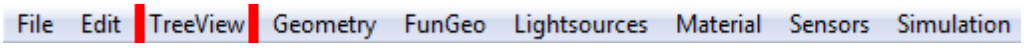


Figure 3.6: Tree View Menu

The following menu entries are available under *TreeView*:

3.2.1. Selections

clear selection	Unselects all selected objects.
execute/edit selection	Executes or edits the selected object.
edit selection by script	Edits the selected object in a new script view.

3.2.2. Tree Node Properties

pick-able property	Enables or disables the object's pick-able attribute, which enables its selection. If set, the object is assigned a "P" in the tree view.
traceable property	Enables or disables the object's traceable attribute , which enables its ability to be ray traced with a shape. If set, the object is assigned a "T" in the tree view.
set property	Shows or hides the geometry in Geo views. If set, the object is assigned a "V" in the tree view.

3.2.3. Tree Display

node details	If set, shows details of the selected tree node.
expand tree	Expands the tree in full detail (making all leaves visible).
collapse tree	Collapses the tree, making only the root node visible.
flatten tree	Makes the basic objects in the tree visible (less than expand, more than collapse).

3.2.4. Elements Display Filtering

elements display filtering	The "Elements Display Filtering" option pops up a dialog with check boxes to toggle the tree display of various LucidShape data types or properties on or off, see the picture right below.

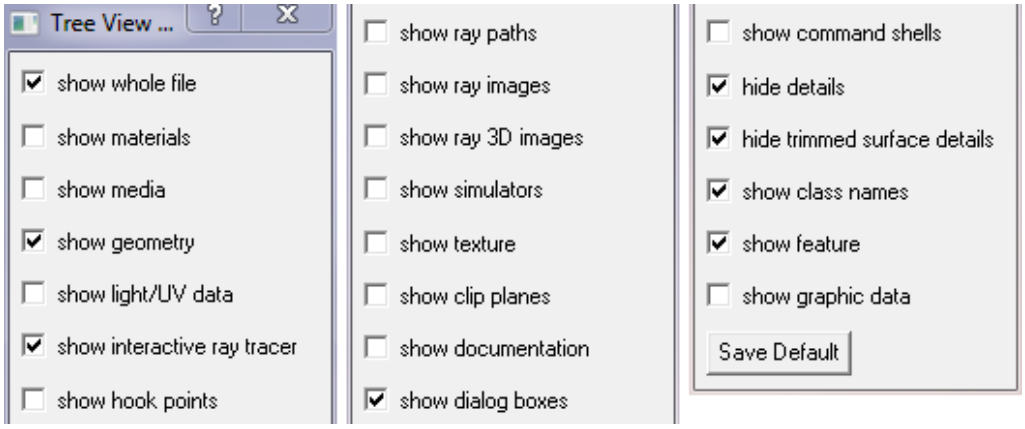


Figure 3.7: The Elements Display Filtering Dialog

3.3. Text View

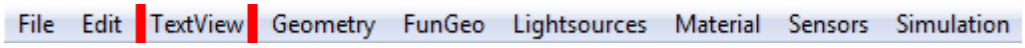


Figure 3.8: Text View Menu

The following menu entries are available under *TextView*:

update	If the text view is the representation of another view, and data have been changed, the file is reinterpreted and all views are updated with the new data from the edited text view.
execute	Executes the current script file, the program output is displayed in the message log view.
set font	Opens a dialog box where one might choose the text font.
syntax highlight	Makes a syntax highlighting.
text background color	Allows the changing of the background color within the text view in focus.

If the text view contains a script file, it is also addressed as "ScriptView". A script is executable within LucidStudio, and therefore differs from the text representation of your experiment displayed in the text view.

3.4. UV View

When you are using the UVDataView, also the usual *Edit* menu is replaced with the *UVEdit* menu. Here two further options for exporting the results are available: Copy Metafile and Copy Bitmap.

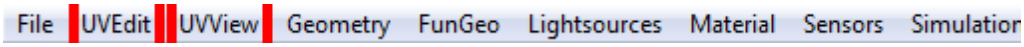


Figure 3.9: UV Data View Menu

The following menu entries are available under *UVView*:

3.4.1. General Commands

Info	Shows information concerning the UV data, e.g. range, minima and maxima. The information is plotted in the message box.
restore from Simulation Result	If pressed the original result from the simulation is restored, so all changes are discarded (like e.g. filtering).
Re-frame	Re-frames the value range to the default range after zoom etc.
Re-frame Selected	Re-frames the value range to the default range after zoom etc.
fit view	Sets the size of the view frame to the ratio of the value rectangle (minimizes the size of the view frame while still showing all of the data).
Duplicate View	Duplicates the view currently selected in focus.
Polar View	Toggles the display type of LID data between "standard" (the default) or "polar". Polar views are especially suitable for Z-type LID data.
Candela	Shows units and axis annotation text for candela.
Cd → Lux on 25m	Shows units and axis annotation text for lux on a R=25m sphere.
Cd → Lux on 10m	Shows units and axis annotation text for lux on a R=10m sphere.

3.4.2. Color Scaling and Scene Lines

all same max/min	All UV data objects of all data views get the same UV value range. The range is taken from the current setting in the data view in focus.
linear/log scale	Toggles between a linear and a logarithmic color mapping.
false gray/false color	Toggles between a gray or false color value display.
only ISO Lines	Toggles between the display types of iso lines (on/off).
isotropic view	Toggles the isotropic view on/off. In an isotropic view the ratio of the display rectangle is the same as the value rectangle.
ISO lines	Toggles iso lines on/off.

color bar	Toggles the display of the color bar.
grid	Toggles the display of the grid.
scene lines	Selects the display types of scene lines, available are: • No Scene Lines • Scene Lines ECE Right Hand Traffic • Scene Lines ECE Left Hand Traffic • Scene Lines Driver • Scene Lines 20m View • Scene Lines Birds Eye

3.4.3. Operations on the Data

Filter

basic smoothing	Performs a smoothing of the uv-data with a basic algorithm.
advanced smoothing	Performs a smoothing of the uv-data with an advanced algorithm.
advanced smoothing dialog	Opens a sub-dialog to adjust parameters for the advanced smoothing algorithm (see picture right below).
iterations	The smoothing algorithm performs several iterations on the given UVData. More iterations take longer but usually lead to better results.
filter weight	Weight of the smoothing term (greater than 0). Greater values in the filter weight field yield in an increased use of the basic filtering algorithm in the advanced filtering of the UV data. So filter weight is the weight of the basic filtering algorithm to be used in the advanced filtering.
edge smoothing	The value of the smoothing of the edges of the beam pattern. The higher the value the less sharp edges will remain (i.e. softening of the cutoff).

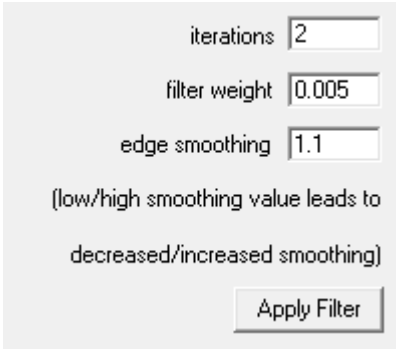


Figure 3.10: The Advanced Smoothing Dialog

Gradients

gradient u	Shows the distribution's gradient in u direction.
gradient v	Shows the distribution's gradient in v direction.
gradient FMVSS108	Shows the distribution's gradient, calculated according to the FMVSS 108 regulation.

Convert

Pole Conversions

convert to X pole	Converts the angular coordinate system of a given light data distribution to the B type (x pole).
convert to Y pole	Converts the angular coordinate system of a given light data distribution to the A type (y pole).
convert to Z pole	Converts the angular coordinate system of a given light data distribution to the C type (z pole).

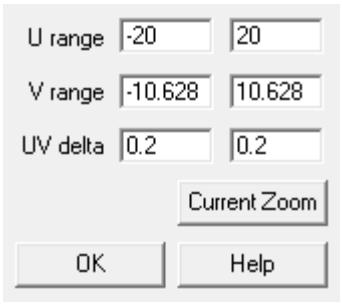


Figure 3.11: UV Data convert to Pole Dialog

U range, V range	The new UV-range of the chosen pole-type.
UV delta	The new UV-delta, i.e. the new entry size.
button: current zoom	Adjusts the values to the current zoom of the LID data to be converted.

Project to Plane

Starting from a new source position, the light distribution is projected onto a new plane with the given xy-range.

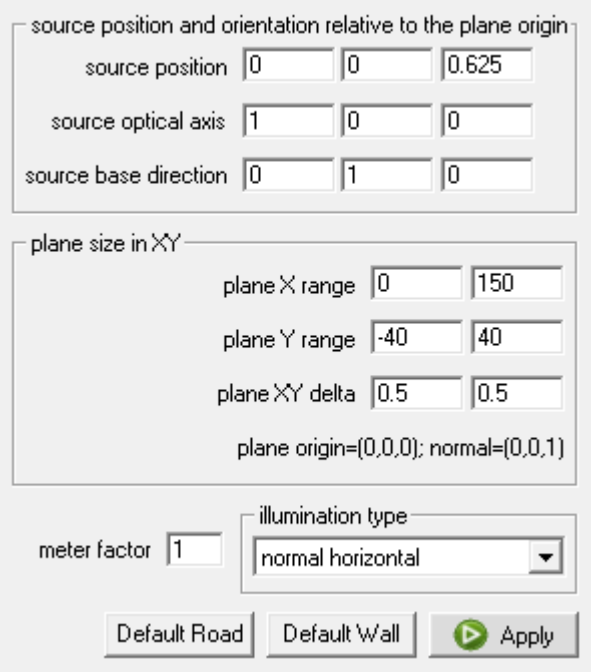


Figure 3.12: UV Data project to Plane Dialog

source position	The new position of the light source.
source optical axis	The direction of the z-axis.
source base direction	The direction of the x-axis.
plane x-range	The u-range of the light distribution.
plane y-range	The v-range of the light distribution.
plane XY delta	The size of each entry in u and v.
meter factor	The meter factor is used to calculate the illumination in [lx] = [lumen/m^2].
illumination type	Allows the user to select between six different types of illumination types such as normal horizontal, radial (no use of a cosine factor), vertical, half spherical, cylindrical, and half cylindrical
button: Default Road	Uses a default road position with the light source at z=0.625.
button: Default Wall	Uses a default wall position with the light source at z=10.

Plane to Pole Conversions

plane → x pole	Calculates a luminous intensity distribution from a illumination on a plane. The given light distribution is reversely projected into a X pole angle system.
plane → Y pole	Like above, but the projection occurs in a Y pole angle system.
plane → Z pole	Like above, but the projection occurs in a Z pole angle system.

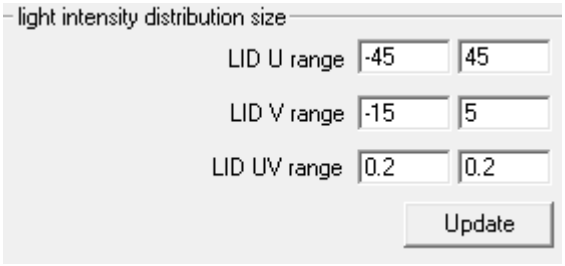


Figure 3.13: The Plane to XYZ Dialog

source position	The position of the light source.
source optical axis	The direction of the Z-axis.
source base direction	The direction of the X-axis.
LID UV range	The new UV-range for the calculated luminous intensity distribution.
LID UV delta	The size of each entry in U and V.
meter factor	The meter factor is used to calculate the illumination in [lx] = [lumen/m^2].
Illumination Type	Allows the user to select between six different types of illumination types such as normal horizontal, radial (no use of a cosine factor), vertical, half spherical, cylindrical, and half cylindrical.
button Default Road	Uses a default road position with the light source at z=0.625.
button Default Wall	Uses a default wall position with the light source at z=10.

Remaining Conversions

split data frames	Splits the 3D spectral data frames into all of the 2D slices that make up the 3D UV Data Plot.
toggle BRDF relative to specular direction	If you have a BRDF as UV data from a file conversion you can toggle whether or not you want it oriented in the specular direction of the material.

Spectral Calculations

sum spectrum	Adds the individual UV data in a spectral simulation. The result is displayed in the data view, this also disables the "Navigate 3D UVW Data" option. See Navigate 3D/4D UVW/UVWT Data.
convert to color	Opens a new dialog which allows to convert the currently selected LID into a colored LID. There are four conversions selectable: Screen RGB, sRGB, CIE 1932 x,y and color temperature.
human eye vision image	Please see the sub-section below.
color data analysis	Opens a new window which states several color informations about the LIDView like the CIE diagram, the color spaces, the SPD, etc. The currently selected position (sensor cell) can be altered via keeping the left mouse button pressed and moving the mouse

	over the LID. Pressing 'convert' will convert the values from the currently selected choice into the other three ones.
spectrum as RGB	Opens a new window that displays the view's current spectrum as an RGB bitmap. (This applies only for 3D data sets, e.g. results of spectral simulations.)

Human Eye Vision Image (HEVI)

The HEVI Tone Mapping is a tool in LucidShape to modify images with both spectral and high dynamic range (HDR) informations. For this modification it takes the special perceiving of the human eye into account. The goal is to achieve an altered image which looks like the human eye would see the unaltered picture.

To access this dialog, first mark a spectral LID. Now open the dialog either via 'Menu → UVView → Spectral Calculations → Human Eye Vision Image' or open the context menu by right-clicking onto the LID and choose the HEVI entry. Pressing *Apply* creates an extra LID, now using the sliders will affect this extra LID only.

ambient brightness	Adjusts the image for different viewing conditions, i.e. conditions, how the tone mapped image is presented. The human eyes are automatically adjusting to different levels of brightness, this slider alters the image to achieve the same perceiving under different conditions. 1 : normal conditions like e.g. a usual illuminated office; 0 : viewing the image in a (partially) darkened room, e.g. with a beamer or monitor
contrast	The fine tuning for the image's contrast. 0 : lower contrast; 1 : higher contrast
chromatic adaption	Allows to influence the level of the local chromatic adaption for the tone mapped image (i.e. a local shift of the white point).
lower/upper cutoff	Removes some outlier pixels from tone mapped image, effectively pixels with a low/high intensity are ignored for scaling the brightness of the output image. This leads to a better contrast of the displayed image. It works as follows: • the pixels of the image are sorted by their luminance in ascending order • depending on the upper cutoff value, a certain percentage of the pixels with maximal brightness are ignored for scaling the output image. E.g. a value of 1% causes the first 1% of the pixels with the highest brightness to be ignored for scaling the output image. • In analogy to the upper case, the given percentage of the pixels with minimal brightness are ignored for scaling the output image. Please note: The given percentage indicates the amount of pixels to be ignored, i.e. the number of entries in the ordered list. It does not affect the brightness' range directly.

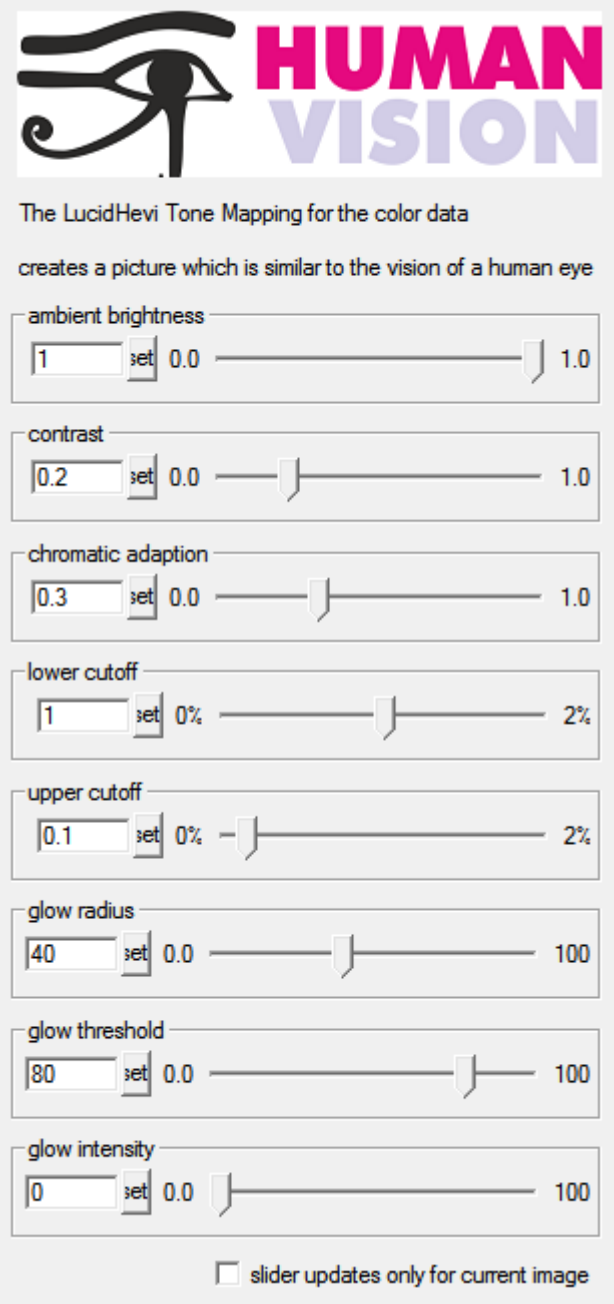


Figure 3.14: The HEVI Tone Mapping Dialog

glow	This adds a glow effect to the tone mapped image. • glow threshold: Selects pixel to be involved into the glow effect. E.g. glow threshold = 80% means that only pixels with a brightness higher than 80% of the maximal brightness will be used for glow. • glow radius: Defines the glow radius around each involved pixel for glow effect (and thus the area to glow). • glow intensity: Scales the intensity of the glow effect.

Resizing and Range Shifting

shift range	Opens a small dialog to enter u and v shift values to shift the data range. Note: If you shift a uv-range with angular coordinates, the range is only shifted and not rotated. If you have a Candela sensor with LID data and shift in order to get a new range, you will get a LID that is different from the LID you get when you start with that new range and make a ray trace. For Candela distributions, you should apply this function only with small shift values.
resize range	A new uv-range $[u_{min},u_{max}] \times [v_{min},v_{max}]$ and uv-resolution (δ_u,δ_v) may be set to the light data view. Affects the selected data only.
rotate	Allows to rotate the uv-data about the x-,y- or z-axis. Rotation angles are specified in degrees. The rotation effect depends on the uv-data type (X, Y or Z)
crop by intensity threshold	Sets all data values which are below a user-specified threshold to zero. By activating an additional check button, the uv-view may also be cropped to the data's effective region (i.e. the bounding rectangle of the set of non-zero data cells).
u-range, v-range	The new uv-range.
uv-delta	The new uv-delta, i.e. the new entry size.
current zoom	Button that works like a reset, the original values are used.

Mirror and Symmetry

mirror in u	Mirrors the data in u-direction.
mirror in v	Mirrors the data in v-direction.
swap u and v	Swaps the u- (horizontal) and v- (vertical) axis.
symmetry in u	Makes the data symmetric in u-direction.
symmetry in v	Makes the data symmetric in v-direction.
symmetry in u and v	Makes the data symmetric both in u- and v-direction.
extent type C to φ=360°	Extents LIDs of type C to a horizontal range of 360°.

Symmetry along u-axis is achieved by averaging the data values of two points each with the same absolute u-coordinate (+/-u) and identical v-coordinates (or vice versa for v-symmetry). The obtained average value is then re-assigned to both coordinates (+u,v / -u,v).

Miscellaneous Operations

Note that the application of the arithmetic operations Plus, Minus, Multiply and Divide requires two data sets which have to be selected by the user. These can be two different beam patterns, for example. The two data sets do not have to be of same size, in which case, the largest data set is the size of the new data set post operation. The application of these operations to scalar values is in any case invalid. For multiplying a data set with a scalar factor, the scale command within this menu can be used instead.

To apply these operations, proceed as follows: 1) Set the first uv-view into focus. 2) Select the operation to be performed. 3) Select the second uv-view to perform the operation. The result of the chosen operation will be directed into the first view that was selected in step 1). In consequence, the original contents of the first view will be replaced with the new result.

LucidStudio uses forward notation, "as written". In case of the operation "Minus" for example, the above 1)2)3) sequence leads to a term of the type first_view - second_view = result_view.

plus	Adds a second uv-view's data to the data of the view in focus (first view).
minus	Subtracts a second uv-view's data from the data of the view in focus (first view).
multiply	Multiplies a second uv-view's data by the data of the view in focus (first view).
divide	Divides the data of the view in focus (first view) by a second uv-view's data.
scale	The light data values are scaled by a factor, to be supplied by the user in a small pop up dialog. Affects the selected data only.
stretch	The light data values are stretched by a horizontal or vertical (positive or negative) factor, to be supplied by the user in a small pop up dialog. Affects the selected data only.
absolute	Replaces all values by their absolutes.
negative to zero	Sets all negative values to zero.
regular cell size	Adapts the cell sizes of the current uv-data view to one common value. Useful to regularize imported IES or CSV LID files which allow irregular cell sizes.

Special Views

The commands in this subsection generate special LID data views from an existing Candela LID distribution. All views are configurable via the respective dialog.

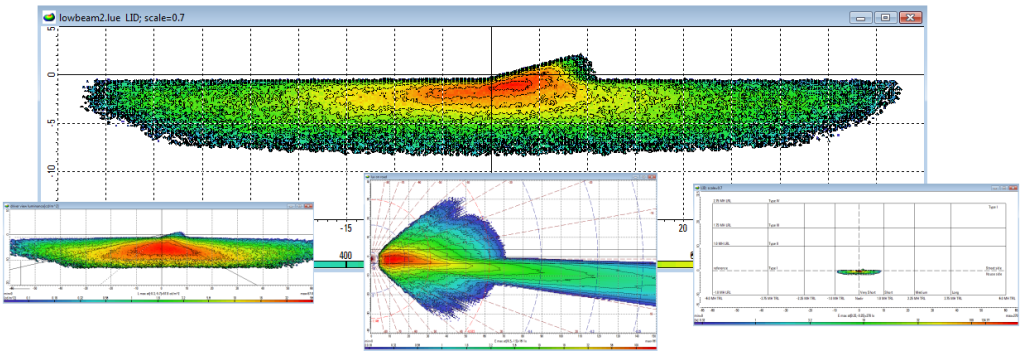


Figure 3.15: A Candela LID from a Low Beam simulation and the special views generated from it (from left to right: driver view, bird's eye view, street luminaire diagram).

Please note: All coordinates in the next two dialogs are the same as in *LucidDrive*: The x-axis points in the driving direction, the y-axis points to the left and z points up. E.g. a source being positioned at (0,-0.8,0.625) is located directly at the front of the car (x=0), a bit to the right (y‹0) and at z=0.625m above the road.

Driver View

The generated LID shows the UVData the driver would see from the driver's position.

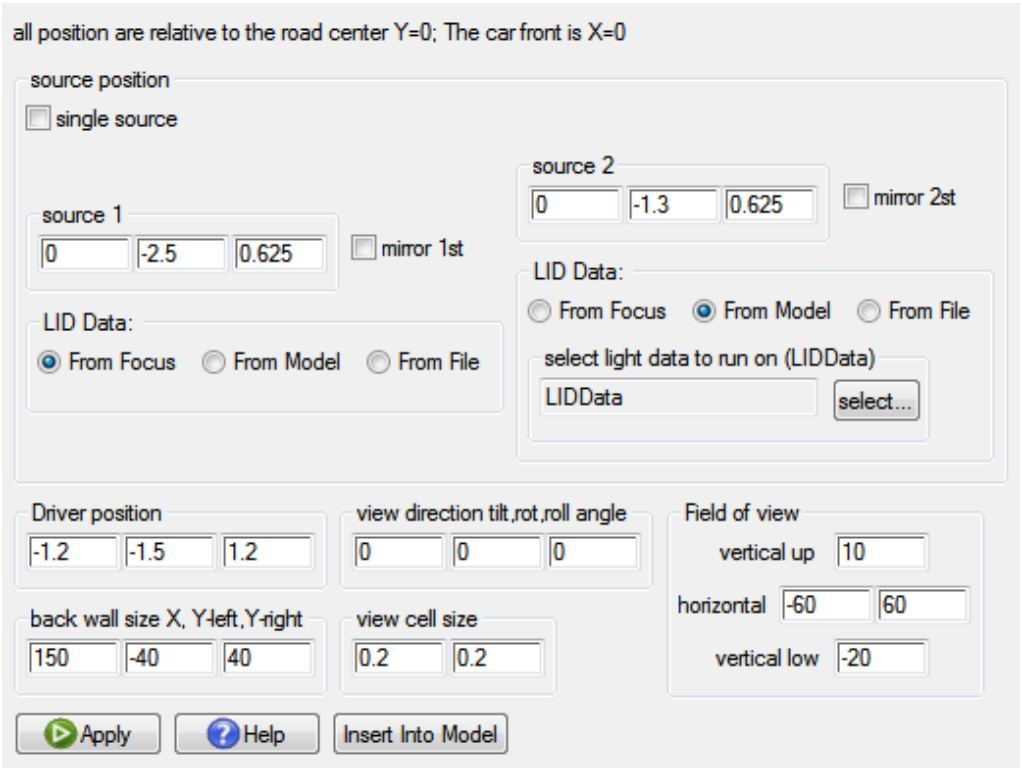


Figure 3.16: The Driver View Dialog

flag: single source	Allows to use either one or two light-sources.
source 1/2	Determines the position of the respective light-source, using the above explained coordinate system.
flag: mirror 1st/2nd	In case of using two light-sources, one them can be mirrored by activating the respective flag, the other LID will be ignored.
radio button: LIDData	The LID to be used can be either an UVView in focus, a LID stored in the TreeView or a LID saved as a file (from left to right).
driver position	Determines the position of the driver.
view direction tilt, rotation, roll angle	Parameters of the view direction relative to the default looking straight ahead.
back wall size x, y-left, y-right	Determines the position of the aiming wall (x-coordinate) and the size via y_{left} and y_{right}.
view cell size	The cell size of the generated view.
field of view	Determines the FoV, i.e. the size of the view to be generated. All four values are given in degree.
button: Insert into model	If pressed the feature is added to the TreeView, i.e. it becomes a part of the model.

Bird's Eye View

The generated LID shows the UVData which would be seen from a position 20m above the road.

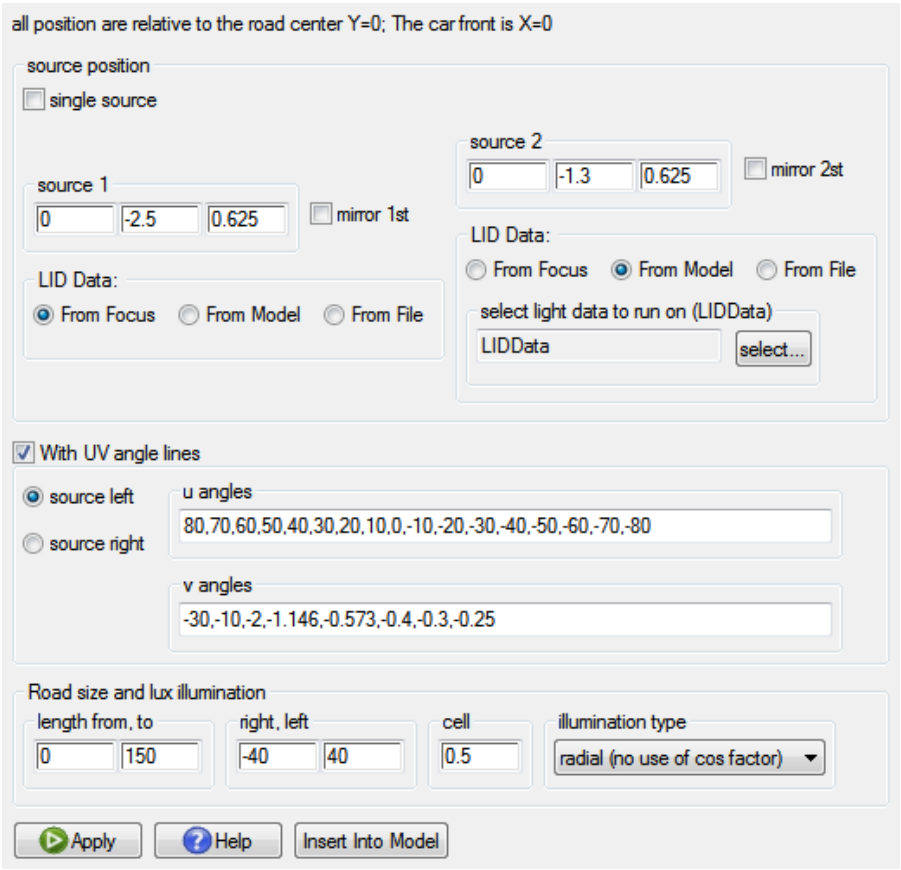


Figure 3.17: The Bird's Eye View Dialog

flag: single source	Allows to use either one or two light-sources.
source 1/2	Determines the position of the respective light-source, using the above explained coordinate system.
flag: mirror 1st/2nd	In case of using two light-sources, one them can be mirrored by activating the respective flag, the other LID will be ignored.
radio button: LIDData	The LID to be used can be either an UVView in focus, a LID stored in the TreeView or a LID saved as a file (from left to right).
flag: with UV angle lines	If activated one can display UV angle lines in the generated view.
radio button: source left/right	These angle lines can be displayed either for the left or the right source.
length from, to	Determines the size of the road via x_{min}, x_{max}.
right, left	determines the road width via y_{right} and y_{left}.
cell	The cell size of the generated view.
illumination type	Allows the user to select between six different types of illumination types such as normal horizontal, radial (no use of a cosine factor), vertical, half spherical, cylindrical, and half cylindrical.
button: Insert into model	If pressed the feature is added to the TreeView, i.e. it becomes a part of the model.

Street Luminaire Diagram

Generates a street luminaire diagram for IES classification. This polar diagram helps to classify roadway and area illumination fixtures ("poles") according to the resulting photometric properties of a mounted light source (LID) on the ground. It uses the distance to the half maximum candela trace and the maximum candela peak as evaluation criteria. Additionally, a illumination diagram with Cartesian coordinates opens up to show the resulting distribution on the road surface with the road lines from the IES diagram.

First, select a LID (UVData View, [cd] distribution) and then start the Street Luminaire Diagram via the context menu (right click on uv-data view).

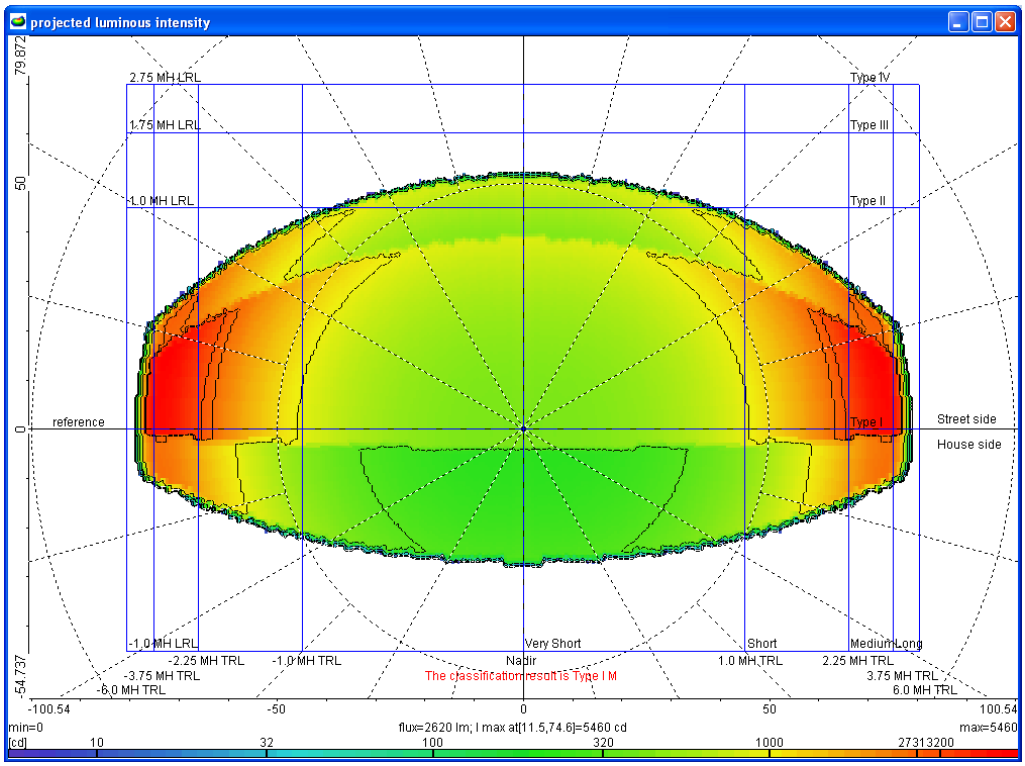


Figure 3.18: The Street Luminaire Diagram: this light is of Type I, medium range (M).

Lateral distances are considered in multiples of the mounting height MH. Longitudinal roadway lines (LRL) and transverse roadway lines (TRL) are used to divide the lateral ranges into zones. The parameters of the Street Luminaire Diagram view are listed below.

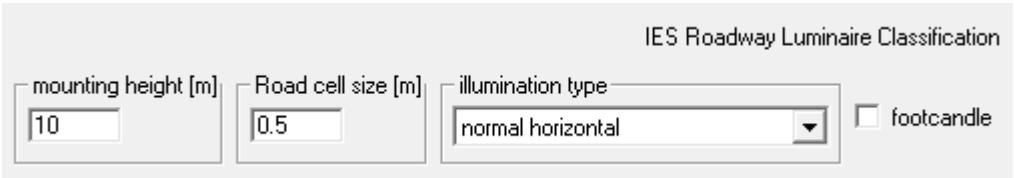


Figure 3.19: The Street Luminaire Diagram Dialog

mountain height (MH) [m]	Vertical position of the light source above road level.
road cell size [m]	Cell size of the road sensor, i.e. the distance between measurement points (standard of 0.5 m).
illumination type	Select between normal (horizontal), radial, vertical, half spherical, and cylindrical illuminance. The illumination is measured at road surface level.
foot candle	Use the unit footcandle instead of lux (1 footcandle is approx. 10.764 lux).

The illumination range along the road is obtained from the diagram. A light is classified to be of (Very) Short, Medium or Large range if the point of maximum intensity [cd] lies within the according range along the road.

As a result of plotting the UVData in the Street Luminaire Diagram, the definition of the roadway luminaire class from the five types I, II, III, IV, and V is done. It characterizes the transverse width of the light distribution across the road (on the street side). For this classification, the half maximum iso intensity trace is only considered within the longitudinal range of the light (S, M, L), as discussed above. The types are briefly characterized below.

Type	Half-maximum candela iso line position / other symmetry.
Type I	Between 1 x MH LRL on house side and 1 x MH LRL on street side.
Type II	On street side partly or entirely between 1 x MH LRL and 1.75 x MH LRL, but not beyond.
Type III	On Street side partly or entirely between 1.75 x MH LRL and 2.75 x MH LRL, but not beyond.
Type IV	On street side partly or entirely beyond 2.75 x MH LRL.
Type V	Symmetry: circular, invariance against z-axis rotation.

3.4.4. Background Colors

The commands in this section open a Windows color selection dialog to select the desired background color. This dialog automatically closes after applying the chosen color by clicking ok.

set background color	Sets the background color of the view in focus.
set default background color	Sets the default background color for uv-views. Any change will first come into effect when opening the next uv-view. It will neither affect the view currently in focus nor other already opened uv-views unless a redraw command is executed subsequently.

3.4.5. Spectral specific options

navigate in 3D/4D data set	Navigates through a 3D/4D UVW or UVWT data field. W and T are additional variables like road curvature or wavelength. The UVData is displayed as a function of additional parameter. That does only works if the data has more parameters than U and V. (see the picture right below)

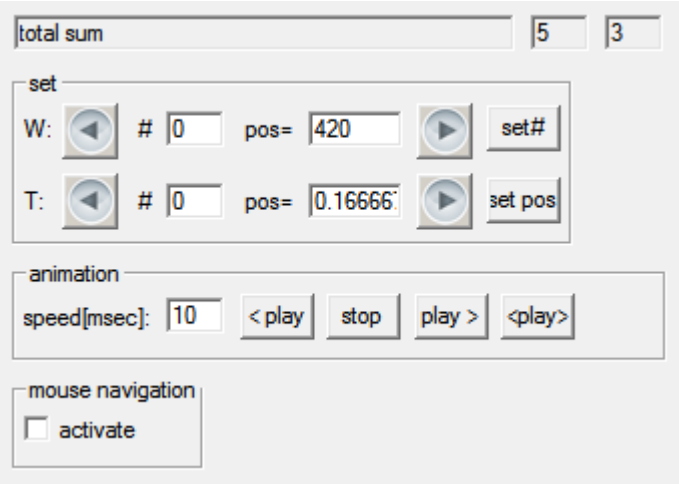


Figure 3.20: The Navigate 4D Data View

#, position	The position within the uv-data.
animation time	The animation time in milliseconds.
‹play, play›	Runs through the uv-data in forward or backward direction.
stop	Stops the animation.
flag: mouse navigation	If checked, one can scroll through the channels by pressing the left mouse button and then moving the mouse left/right or forward/backward.

3.4.6. Detailed View Properties

The last entry in this menu *Detailed View Properties* brings up a dialog for editing data display properties. The dialog changes dynamically depending on the type of the LID marked. In case of a LID with spectral informations (i.e. single wavelength channels) results in the usual dialog as shown just below, color informations however lead to a different dialog shown in the third picture below.

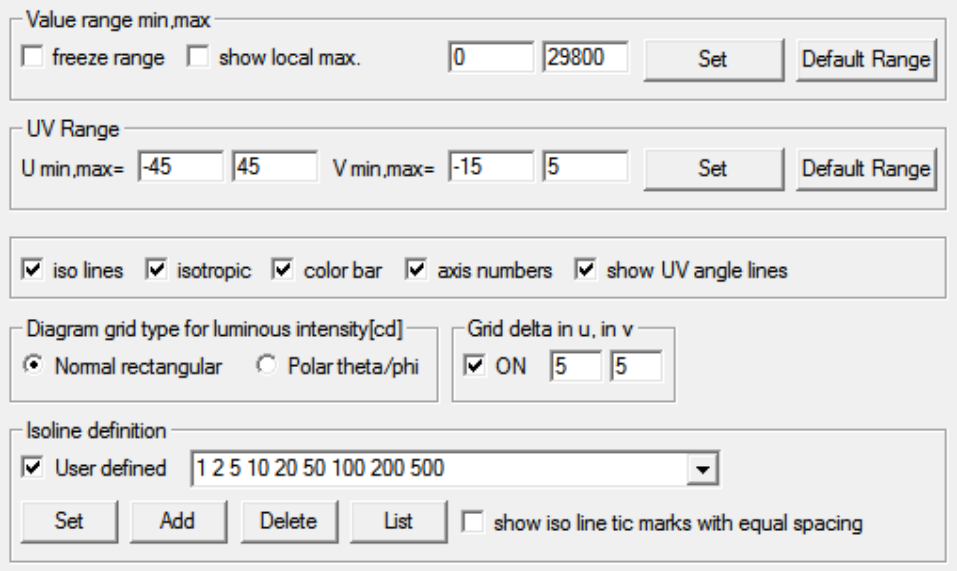


Figure 3.21: The UV Data Properties View (I)

flag: freeze range	If checked, the value range in the UVView is not updated after every simulation, but stays in the specified range.
flag: show local maximum	If checked, the UVView shows additional informations in the text line.
value range min, max	Sets the minimum and maximum value.
button: set	If pressed, the current settings are applied onto the UVView.
button: default range	If pressed, the value range in the dialog is adopted from the values in the UVView.
uv-range	Determines the range of the uv-data via u_{min}, u_{max}, v_{min} and v_{max}.
flag: iso lines	Switches the iso lines (=lines with same intensity) on/off.
flag: isotropic	Switches isotropic on/off. If disabled, the view will be distorted in such a way that the model's bounding box fills the whole view. This is convenient for models with extreme aspect ratios, like for example certain spectral curves with a large wavelength range and a small value range. If enabled, i.e. in an isotropic view, the ratio of the display rectangle is the same as the value rectangle.
flag: color bar	The color bar at the bottom of the data view may be switched on/off.
flag: axis numbers	Switches the axis legend on/off.
grid delta in u, v	Determines the resolution of the grid, the flag allows to show/hide the grid lines in the UVView.
show UV angle lines	If checked, the UV angle lines in a polar view are shown, otherwise they are hidden.
diagram grid type	Toggles the view and the grid between a rectangular and a polar view.
flag: grid delta	If checked the gap of the grid lines can be specified independently for u and v.

flag: iso line definition	If checked, sets of iso lines can be chosen, entered or altered. (All sets are saved in the .ini file of the current LucidShape version, here they can be altered too.
button: set	If pressed, the currently active set is applied onto the UVView.
button: add	After been set as active, the iso line set can be saved.
button: delete	If pressed, the currently active set will be deleted.
button: list	If pressed, all saved sets are listed in the MessageView.
flag: show iso line tic marks with equal spacing	If checked, these marks in the color bar are not arranged depending on their respective values, but in equal distances and the color bar is set to white.

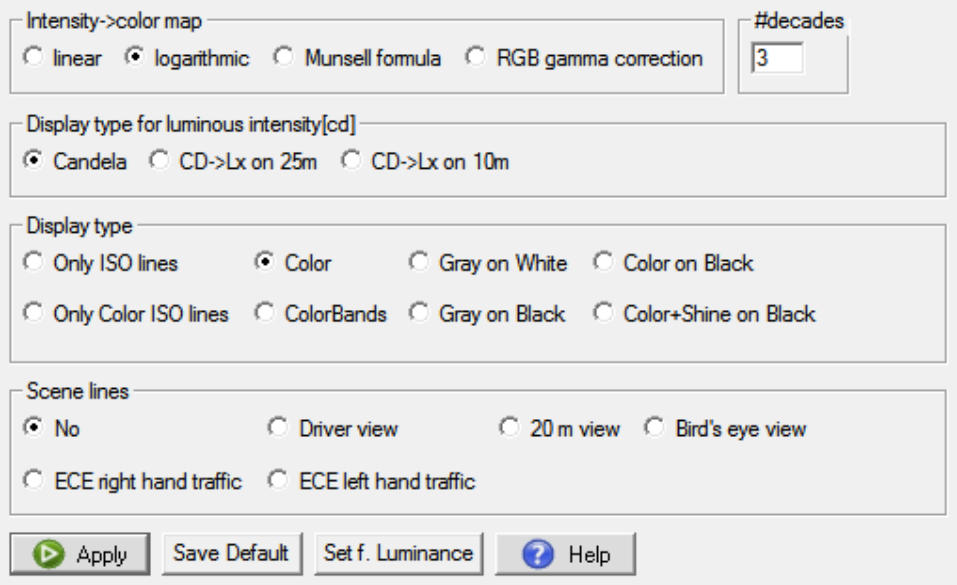


Figure 3.22: The UV Data Properties View (II)

radio button: intensity › color map	This radio button toggles between different modi for the color scaling: Linear, logarithmic, Munsell formula or RGB gamma correction.
# decades	This determines the scaling of the color in the UVView. The general formula is: log(R·10^dec)/log(10^dec); *R* is one channel (RGB) and *dec* the given number of decades.
display type for [cd]	The display type for a candela intensity can be chosen among Candela, Lux on the road at 25m distance and Lux on the road at 10m distance.
display type for [lx]	The display type for a lux intensity can be chosen among [lx] and [fc] (footcandle).
radio button: display type	• *Only ISO lines:* Displays black ISO-lines of a light distribution. • *Color:* Colored display. • *Gray on white:* The light distribution is displayed in gray on a white background.

	• *Only color ISO lines:* Displays colored ISO-lines of a light distribution. • *Color Bands:* Displays color bands. • *Gray on black:* The light distribution is displayed in gray on a black background.
radio button: scene lines	Different types of lines can be added like ECE cutoff or road lines discussed earlier in the UVView Menu.

In case the LID does contain color informations, there are some additional parameters to adjust the view. How can the simulation result - stored in the sensor - be shown on the monitor? The monitor can display 256 channels per color (red, green, blue). That means that the LIDData (already containing color informations) has to be transformed or mapped into these three channels with a range of [0,255] each.

In LucidShape this can be done by choosing among five different types of tone mapping algorithms. They all map HDR Image data (either from LIDData or an HDR Image-format like OpenEXR) into the range of low dynamic display devices (Monitors, Printer etc.). There are three simple mapping algorithms (linear, logarithmic and Munsell) and two more advanced ones (OpenEXR and Reinhard '05):

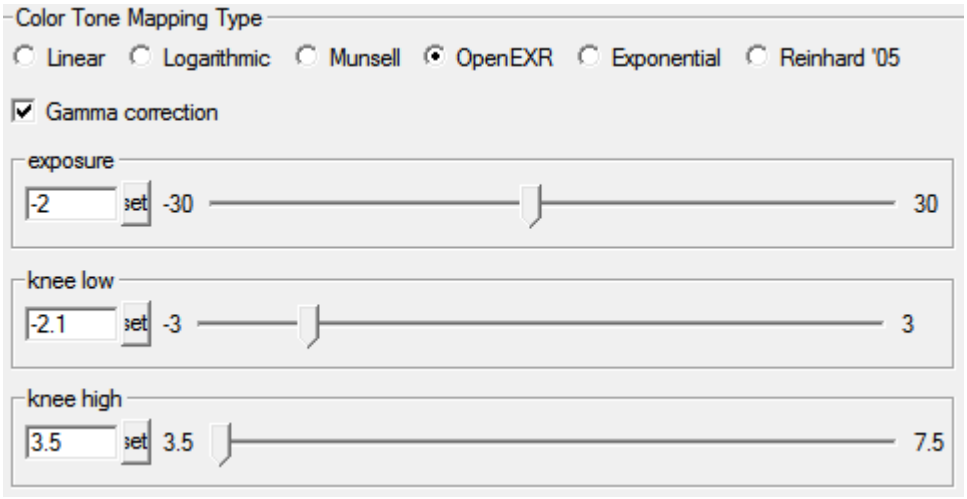


Figure 3.23: The UV Data Properties View (III)

radio button: color tone mapping type	• linear: The mapping into [0,255] is done linearly, this is the most primitive type. • logarithmic: The mapping is done logarithmically, this is much closer to the human perception. • Munsell: This mapping version also belongs to the class of logarithmic mappings, it will produce a smoother solution than the logarithmic mapping. • OpenEXR/exponential: The intention of these two types is to emulate camera settings. The original mapping algorithm suggested by OpenEXR was introduced to post process video- and photo-shots. They both belong to the family of logarithmic algorithms, too. In the exponential version one is able to modify

	the slope of the mapping curve and the mapping is scaled to the incoming data. • Reinhard '05: This type uses the Reinhard/Devlin method. The mapping to low dynamic display devices is similar to adaptation processes which occur in the human visual system. Physiological evidence suggests that adaptation already occurs in the photoreceptors, leading to a straightforward model that can be easily adapted for tone reproduction. This type tries to emulate the adaptation processes which occur in the human visual system.
flag: Gamma correction	Activating or de-activating the gamma correction performs a change of the color space to be used.
clip RGB values (linear, logarithmic, Munsell, exponential)	Available for the four types linear, logarithmic, Munsell and exponential. The clipping determines the maximal luminance value, so this luminance and all higher values are mapped to 255. This is independent to the used type, i.e. it has no effect onto the form of the mapping for the data below this value.
exposure (OpenEXR)	This parameters effects the brightness of the shown picture. It is scaled in f-stops like a camera. A change of +1 (-1) doubles (halves) the shown brightness.

3.5. Light Screen View

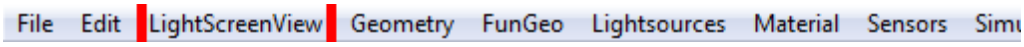


Figure 3.24: Screen View Menu

The following menu entries are available under *LightScreenView*:

3.5.1. General Commands

info	Shows information about the model in the message box.
zoom in/out	Zooms in/out in small steps, you can also zoom in/out by moving the mouse wheel in upper/lower direction or by using the SHIFT and the LMB while moving the mouse up/down.

3.5.2. Display Commands

XY View	Sets the standard XY view.
YX View	Sets the YX view, i.e. reverses the standard XY view.
X Pole Screen	Sets the display to x pole angles.
Y Pole Screen	Sets the display to y pole angles.
z-plane screen	Sets the display to geometric units at the distance of the plane in the z-axis (used mostly for near-field applications where units are used rather than angles).

ECE Lines	Toggles the display of ECE road lines on/off.

3.5.3. Coloring and Display Properties

The first three commands in this section open a Windows color selection dialog to select the desired background color.

background color	Sets the background color of the view in focus.
default background color	Sets the default background color for all light screen views. The change will first come into effect when opening the next light screen view. It will neither affect the view currently in focus nor other already opened views unless a redraw command is executed subsequently.
black & white	Toggles between black & white and color display.

3.5.4. Screen View Properties

This selection opens a property sheet to set various advanced light screen view options.

screen type	Lets you choose between x-pole or y-pole angle display. The z-plane display changes from angles to geometric units in the display at the z-plane position.
plane distance	The distance of the projection plane (only for the z-plane option).
with ECE road lines	Enables the display of ECE road lines in the screen view (only for x- and y-angles).
grid delta xy	Sets the resolution of the displayed grid (only for x- and y-angles).
text position and range	Sets the text position and range (only for x- and y-angles).
position of grid numbers	Two check boxes which determine the positions of the horizontal (bottom or top) and vertical (left or right) axes annotations.

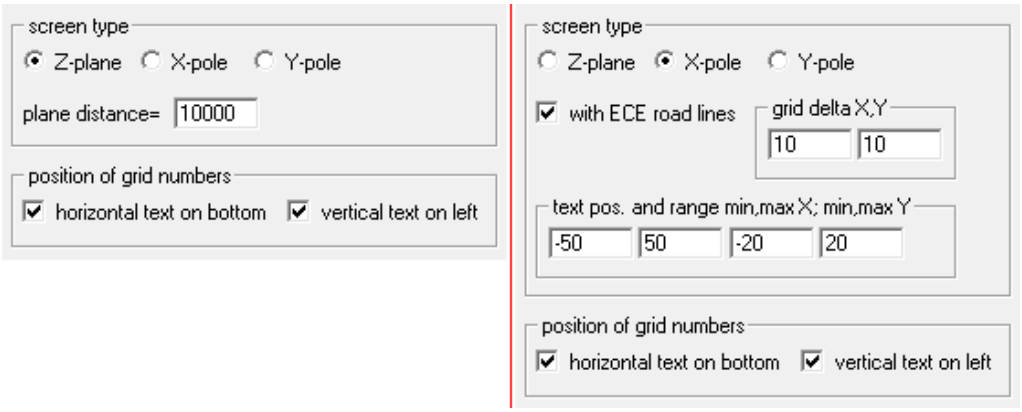


Figure 3.25: Screen View Menu

Chapter 4. Geometry

The better part of a design process is typically spent with development or optimization of the detailed geometrical aspects of an optical system. Consequently, the LucidStudio GUI offers you a variety of commands to create, modify and manage geometrical objects. The Modify commands, for example, include operations to generate, split, merge and in any other way alter mesh, Bézier and NURBS curves and surfaces.

Besides the necessary standard operations to create 3D geometrical objects from mathematical functions or out of other datasets, several pre-installed functions guide you through the creation of typical optical geometries like aspherical lenses or parabolic surfaces. Most of these commands lead you to a dialog that will automatically offer you to also choose a material, which will be assigned to the geometry you are currently building (see for example Section 4.5, “Standard Surfaces”). You can therefore choose between the three material classes: actors, light sources or sensors. Thus, a complete ready-to-use shape can be created instantly. This allows you to quickly create the elements of your experiment without time-consuming multi-step procedures. The details of material properties are treated in the chapters on *Light Sources, Materials* and *Sensors*.

When creating a new geometry, the local axis system is also an orthonormal x,y,z system. However, you are free to define the actual orientation of this local system with respect to the global axes of the experiment and thus orientate the new object upon its creation.

Some commands, like the group of Procedural Surfaces, requires the selection of one or more existing geometries as parameters. To create an extruded surface, for example, you choose a curve to be translated along a vector you will also have to define. After that you may create the final surface geometry from these parameters.

Note, that some of the dialogs that you will use more often, e.g. the Transform dialog, are suited to be left open on your screen, so you can access them again for repeated application of the same command. This way you do not have to re-open the same dialog again and again for each object. If you choose "selected objects" instead of "all objects" in a dialog, it will affect only the shapes or geometries you have selected when you press the apply button.

Almost every object in LucidShape has four "Attributes" (or flags) : Editable (E), Visible (V), Pickable (P) and Traceable (T). All four flags can be toggled pressing Ctrl+Shift+Letter, just mark a shape, highlight either the Geo- or the TreeView and press the respective hot key.

"Editable" is kind of a protection against accidental changes. If unchecked, the respective shape can be examined (to see the options and parameters), but it cannot be altered. To change the values just enable the flag "E" and open the edit dialog as usual.

"Visible" determines whether this object is visible in the geometry view or not (used to enhance the clarity in more complex models). If "Pickable" is checked the respective object is pick-able in the geometry view, this can be set off to make it easier to mark and grab single objects. The fourth attribute is "Traceable". If checked this object takes part in the light simulation, if unchecked it is not traced in the ray-trace process (kind of 'invisible' for the rays, not to be mismatched with 'V' in the geometry view!). Every single attribute can be set independently to the other three, if an object is neither visible nor pick-able it still takes part in the light simulation as long as it is traceable.

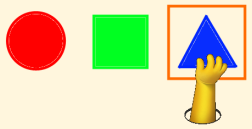
A detailed description of the commands and their required parameters for valid execution are listed in the following sections of this chapter.

4.1. Basic Operations on Geometric Objects

This section presents basic operations on geometry like selecting, modifying, converting etc.

4.1.1. Selection, Deletion and Renaming

These options allow the selection or deletion of objects via several filter parameters which are listed in the subsequent table. These commands are available under 'Geometry → Select' and 'Geometry → Erase' respective. You may enter the type name in the dialog box. If you enter for instance "PlanarSurface", all planes will be selected.

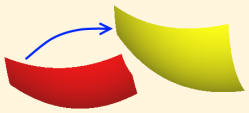


"Make Unique Names" generates and assigns unique identifiers to all objects in the model. This command is available under 'Geometry → Make Unique Names'.

by type name	Selects or deletes all objects with the assigned type name (to be entered in a small dialog). If you enter for instance "PlanarSurface", all planes will be selected or deleted.
by name	Selects or deletes all objects with the assigned name (to be entered in a small dialog). If you enter for instance "surface16", all objects with that identifier will be selected or deleted.
by color	Selects or deletes all objects with the assigned color (to be entered in a small dialog as 3 numbers representing RGB values). If you enter for instance (255,0,0), all red objects will be selected or deleted. Alternatively, the user may enter the selection color indirectly by selecting one object; the object's color will be taken as filter in that case.
by EVPT attribute	Selects or deletes all objects with the chosen flag combination.
all	Selects or deletes all objects in the scene.
all shapes, actor shapes, light sources, sensors, ray path bundles, surfaces, NURBS surfaces, mesh surfaces, curves, NURBS curves, polylines, axis systems, points	Selects or deletes all objects of the selected type in the scene.
make unique names	Generates and assigns unique identifiers to all objects in the model.

4.1.2. Transform

This dialog allows to move, to rotate, to scale, to mirror or copy the selected (or all) objects. The following table lists the essential parts of this tool's dialog.



Most of the tabs do have the possibility of interactive changes with the arrows, the respective amount can be specified via the entries in the respective line, the direct change can be done via the values in the lower part and *Apply*.

Identical to all tabs is the possibility to copy the currently selected object just by a click onto *Copy*.

direct interaction	Moves, rotates, or scales the selected object interactively in the GeoView corresponding to the entered values dx, dy, dz.
move	Shifts the selected object directly by the entered vector.
pivot point x,y,z	The coordinates of the point the geometry is rotated or scaled about.
scale	The x,y,z factors for scaling objects (e.g. a value of 1.13 increases the object in the respective direction by 13%).

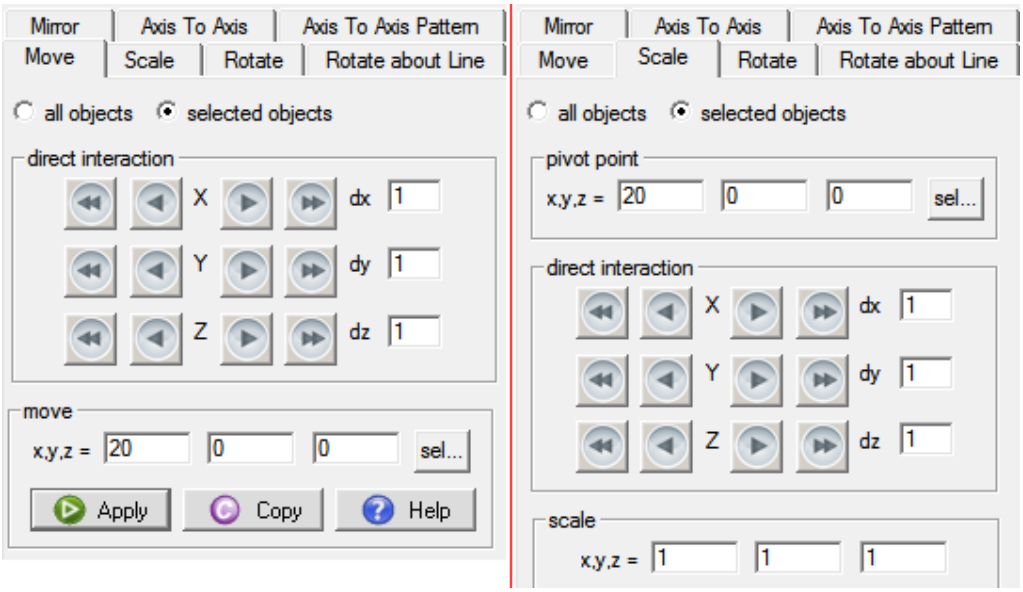


Figure 4.1: The Dialog Parts for Move and Scale

rotate	The direct interaction rotates the object about its own axis system. Using the rotation angles it will be rotated relative to the pivot point. (all values are in degree)
rotate about line	The rotation line defines a straight line running from x,y,z to x',y',z'. The points or the whole line may also be selected.

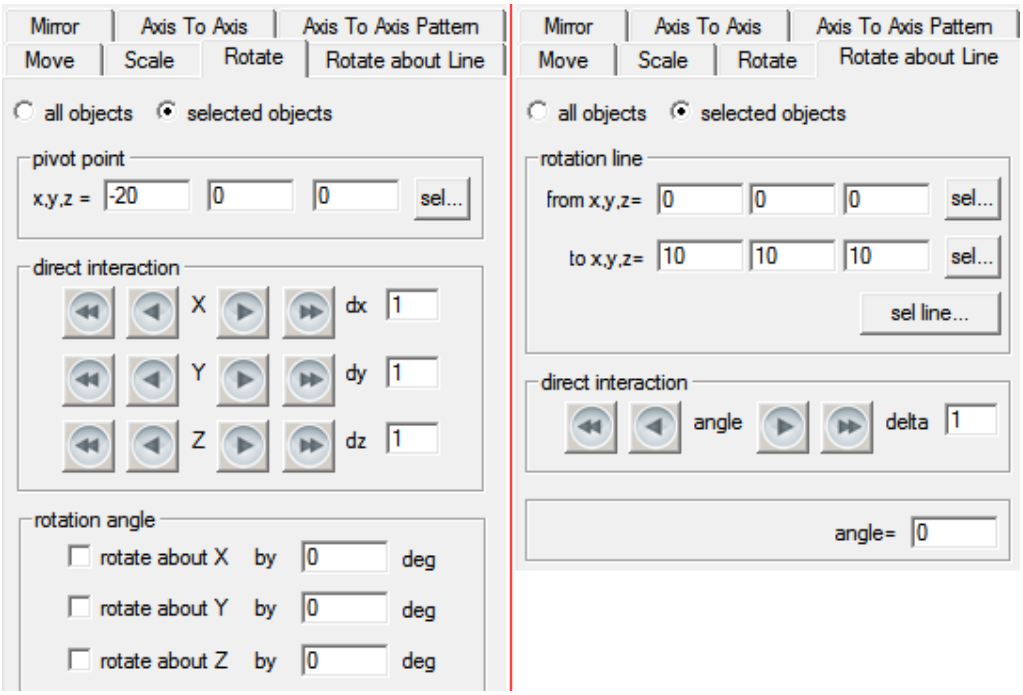


Figure 4.2: The Dialog Parts for 'Rotate' and 'Rotate about Line'

mirror	Reflects the selected object about the x-, y- or z-axis.
axis to axis	The object is moved to the chosen axis system (with or without the offset).
axis to axis pattern	Here the object is applied (and copied) onto every axis system in this axis pattern (with or without the offset).
flag: with from axis	If unchecked, the selected object is moved to the chosen axis system. It is keeping its offset (i.e. the global coordinates), now relative to the destination axis system. If checked, the selected object is moved directly from the object axis system (upper selection) to the destination axis system (lower selection) without the offset.

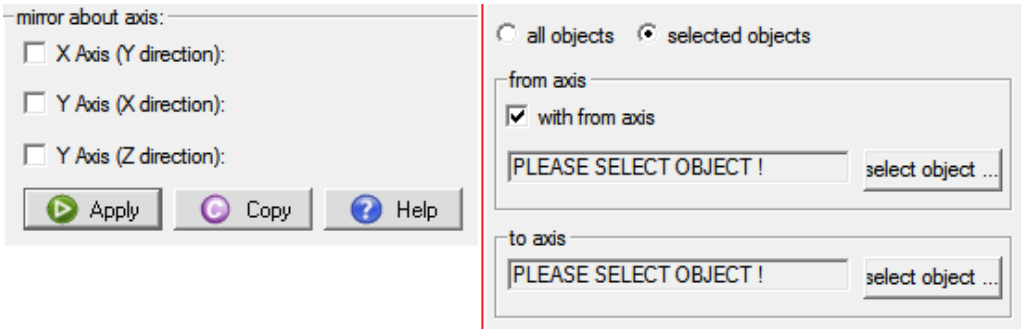
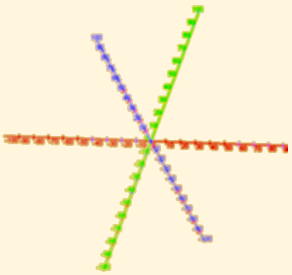


Figure 4.3: The Dialogs for 'Mirror', 'Axis to Axis' and 'Axis to Axis Pattern'

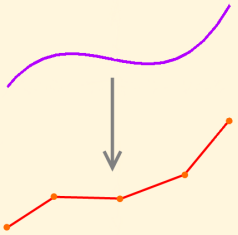
4.1.3. Set Global Axis

This option sets the selected axis system as the global axis system. All space objects will be transformed relative to the position and orientation of that new global axis system.



4.1.4. Conversion of Curves and Surfaces

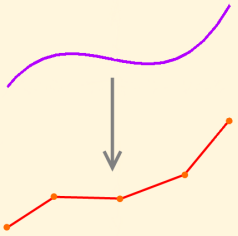
The tools under 'Geometry → Convert' supply conversion between various curve and surface formats.



to NURBS Surfaces	Converts either the selected or all surfaces in the geometry view to NURBS surfaces.
to Mesh Surfaces	Tessellates either the selected or all surfaces to triangular Mesh surfaces.
to Bézier Patches	Converts either the selected or all surfaces to Bézier patches. This conversion can only be performed for untrimmed surfaces.
to NURBS Curves	Converts either the selected or all curves in the geometry view to NURBS curves.
to Polylines	Discretises either the selected or all curves in the geometry view to polylines.
to Bézier Arcs	Converts either the selected or all curves in the geometry view to Bézier arcs.

4.1.5. Modify Data Structures of Geometric Objects

The tools under 'Geometry → Modify' allow to modify internal data structures of curve or surface objects. The use of the tools in this section requires some basic knowledge about NURBS curve and surface mathematics.



extract Surfaces	Extracts the surfaces from a selected SurfaceShape or a MultiSurfaceShape.
untrim Surfaces	Replaces trimmed surfaces in shapes or trimmed surfaces by their base surfaces, i.e. removes the trim bounds.
elevate degree of NURBS curve	Elevates the polynomial degree of the selected NURBS curve by the entered value, without changing the curve's geometry.
elevate degree of NURBS surface	Elevates the polynomial degrees in u and v of the selected NURBS surface by the entered values, without changing the surface's geometry.
reduce degree of NURBS curve	Reduces the polynomial degree of the selected NURBS curve by the entered value, without changing the curve's geometry.
reduce degree of NURBS surface	Reduces the polynomial degrees in u and v of the selected NURBS surface by the entered values, without changing the surface's geometry.
modify knot Vector in NURBS curve	Modifies the number of knots to the curve's parameterization knot vector, without changing the curve's geometry. Used for gaining easier control over the curve's Bézier arc components.
modify knot vector in NURBS surface	Modifies the number of knots to the surface's parameterization u and v knot vectors, without changing the surface's geometry. Used for gaining easier control over the surface's Bézier patch components.
swap u- and v-axis on NURBS surface	Swaps the parameters u and v on either all or selected NURBS surfaces: Replaces the NURBS surface function f(u,v) by f(v,u).
invert u-axis on NURBS surface	Inverts the u-axis on either all or selected NURBS surfaces: Replaces the NURBS surface function f(u,v) by $f(u_{min}+u_{max}-u,v)$, where (u_{min},u_{max}) denotes the surface's u parameter range.
invert v-axis on NURBS surface	Inverts the v-axis on either all or selected NURBS surfaces: Replaces the NURBS surface function f(u,v) by $f(u,v_{min}+v_{max}-v)$, where (v_{min},v_{max}) denotes the surface's v parameter range.
auto Swap UV ‹=› XY of NURBS surface	Performs the swap or invert operation automatically in order to create a surface with a positively oriented normal (i.e. df/du * df/dv has a positive sign). Makes only sense for surfaces with non-vanishing tangents (i.e. df/du and df/dv non-zero).
invert NURBS curve	Inverts the parametrization of a curve, i.e. swaps u in -u and vice versa.
match (+merge) two NURBS curves/Surfaces	Makes the user select two NURBS curves/surfaces in a dialog and tries to merge them to one NURBS curve/surface, additionally the old objects can be deleted. This only works if the two surfaces have the same polynomial degree, furthermore the respective parametrizations of the two surfaces have to be suitable for the merge. (Speaking of the border curves of the surfaces, e.g. it is only possible to merge the northern curve of the bottom surface to the southern curve of the top surface, it is not possible to have a connection between two north curves.)
merge several curves	Merges either the selected or all curves into one curve.

close curve; match start+end	Performs a closure for the selected or all curves. Please note: No additional piece of curve will being created, only the first and the last segment will be deformed.
subrange of NURBS surface	Makes the user select a NURBS surface, then selects a sub-range of it via two parameters.
subrange of NURBS curve	Makes the user select a NURBS curve, then selects a sub-range of it via two parameters.
rebuild NURBS curve	Rebuilds the curve via sampling and approximation: The given curve is first converted into a PointRow. Then a fitting curve is created by using this PointRow. This fitting curve is created as a NURBS Curve. Note that the rebuilt curve will most likely not have the exact shape as the original curve.
rebuild NURBS surface	Rebuilds the surface via sampling and approximation: First a point grid is created on the given surface. Then a fitting surface is created by using this Point-grid. The fitting surface is created as a NURBS Surface. Note that the rebuilt surface will most likely not have the exact shape as the original surface.
invert Mesh facets	Makes the user select a mesh (or all) which will be inverted (swaps front and back surface of it).
join Mesh facets	Makes the user select all or just two meshes which will be joined, possibly double points can be deleted.

4.2. Standard Geometries

This section describes the commands that are available under the menu entry *Standard Geometry*, offering the user a number of basic geometric creation tools.

All commands start a dialog which merely consists of three parts:

1. A dialog part controlling general parameters like the name, the color and its local axis system. See the following picture for an example.
2. A dialog part containing entries specific for the geometry to be created, like the focal length of a paraboloid or the radius of a sphere. These parameters are described in the table below.
3. With pressing *Create* this object will be created and shows up in the respective views (e.g. like TreeView, GeoView).

The first and third dialog parts are nearly the same for all geometries. Containing both a geometry and a material the object will become a shape. The *Assign Material* is described in detail in Chapter 7, *Emitter, Actor and Sensor Materials*.

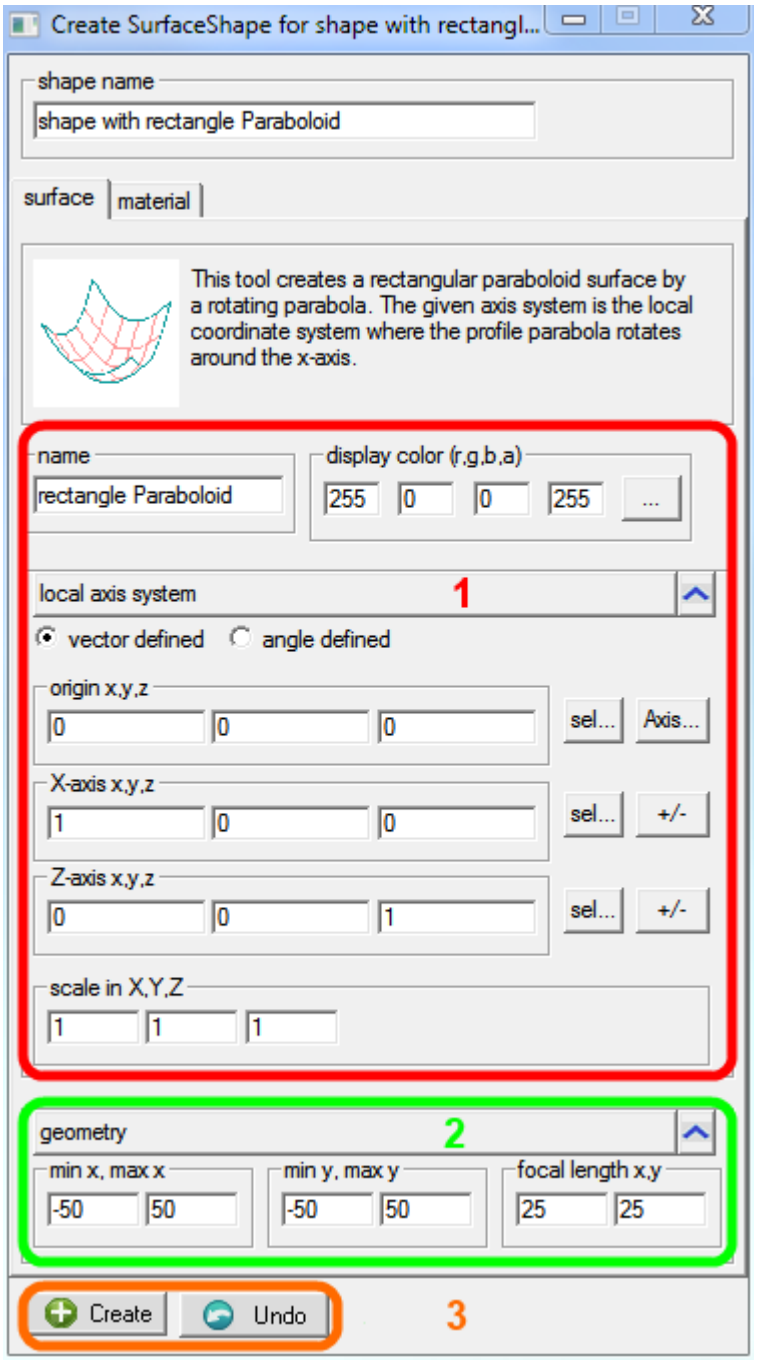
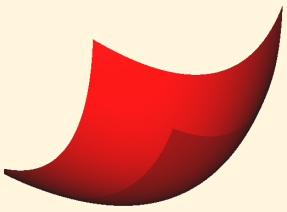


Figure 4.4: The rectangular paraboloid dialog with its three parts: 1) non-geometric entries (common to all dialogs in this section); 2) geometric entries and 3) dialog window options.

4.2.1. Rectangular Paraboloid

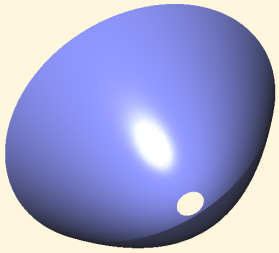
Creates a rectangular paraboloid. The given axis system is the local coordinate system of the rectangular paraboloid. The axis origin is the paraboloid's vertex point. If the two focal lengths differ from each other, the cross-section will become an ellipse. The surface is restricted by the given xy-range.



min x,y, max x,y	The rectangular xy-range in the local coordinate system.
focal length x,y	The focal length(s) of the 2D parabolic curvatures in x- and y-direction.

4.2.2. Rotational Paraboloid

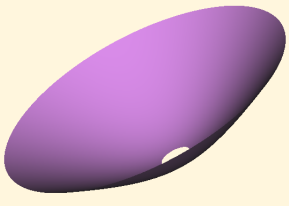
A paraboloid surface is created by a rotating parabola. The given axis system is the local coordinate system where the profile rotates around the z-axis. The axis origin is the vertex point of the parabola. The parabola is defined by a focal length and a pair of start and end radii. The surface may be restricted by a start angle and a sweep angle in the local xy-plane. All angles are defined in degrees and are relative to the local x-axis.



focal length	The focal length of the parabola.
radius min, max	The radius range (inner and outer radii) of the paraboloid.
angle range start, sweep	The start and sweep angle around the rotation z-axis.

4.2.3. Rotational Hyperboloid

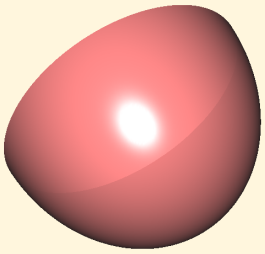
A hyperbolic surface is created by a rotating hyperbola. The given axis system is the local coordinate system where the profile hyperbola rotates around the z-axis. The axis origin is the hyperbola's vertex point. The hyperbola is defined by two focal lengths and a pair of start and end radii. The surface may be restricted by a start and a sweep angle in the local xy-plane. All angles are defined in degrees and are relative to the local x-axis.



focal, virtual focal length	The focal length of the hyperbola.
radius min, max	The radius range (inner and outer radii) of the hyperboloid.
angle range start, sweep	The start and sweep angle around the rotation z-axis.

4.2.4. Rotational Ellipsoid

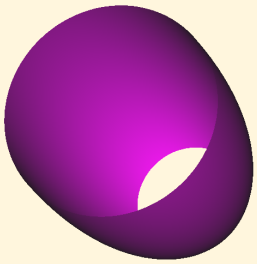
An elliptical surface is created by a rotating ellipse. The given axis system is the local coordinate system where the profile ellipse rotates around the z-axis. The axis origin is the ellipse's vertex point. The ellipse is defined by two focal lengths and an ellipse start and sweep angle. The surface may be retricted by an angle range start and a sweep in the local xy-plane. All angles are defined in degrees and are relative to the local x-axis.



focal length 1,2	The two focal lengths of the ellipse.
ellipse start, sweep	The ellipse's start end sweep angle (0..180), defining the (open) diameters of the ellipsoid at either end around z-axis.
angle range start, sweep	The start and sweep angle around the rotation z-axis (0..360).

4.2.5. Compound Parabolic Concentrator

A compound parabolic concentrator (CPC) can be thought of as a parabolic toroid, i.e. a parabola revolved about an axis other than its axis of symmetry.



entry radius	The entry radius of the CPC.
exit angle	The exit angle for the light rays.

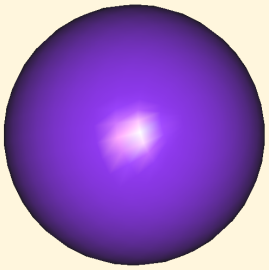
4.2.6. Cylinder

This option creates an elliptical cylinder within the given axis system. The cylinder ellipse (or circle) is orientated in the xy-plane of the axis system. The cylinder is extruded into the z-direction. The length is defined by a start and an end distance in Z. The size of the ellipse is set by 2 radii. The cylinder may be restricted by a start angle and a sweep angle in the local XY plane. The angles are defined in degrees and are relative to the local x-axis.

length from, to	The start and end length along the local z-axis.
radii	The ellipse radii of the cylinder.
angle range start, sweep	The start and sweep angle around the rotation z-axis.

4.2.7. Sphere

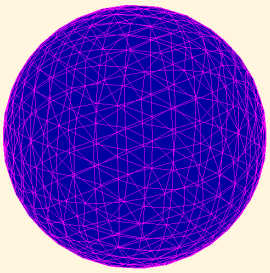
Creates a (sectioned) sphere within the given axis system. The size of the sphere is set by the given radius. The sphere may be sectioned by polar (θ) and azimuth (ϕ) angle sweeps. All angles are defined in degrees.



radius	The radius of the sphere.
θ *start, sweep*	The start and sweep polar angle against the rotation z-axis.
ϕ *start, sweep*	The start and sweep azimuth angle around the rotation z-axis.

4.2.8. Mesh Sphere

Creates a (complete) mesh sphere within the given axis system. The size of the sphere, the color and the mesh density can be set with the dialog box below. This sphere can not be sectioned at creation.



radius	Defines the sphere radius.
mesh density	The mesh density of the surface, the larger this number, the higher the density.

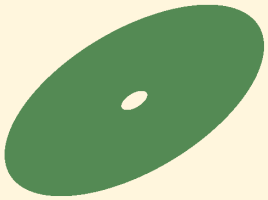
4.2.9. Plane

Creates a plane surface within the XY plane of the given axis system. The plane has its origin in the axis origin. The surface on the plane is restricted to the given uv-rectangle.

min x,y, max x,y	These four values define the uv-range of the rectangular plane section.

4.2.10. Disk

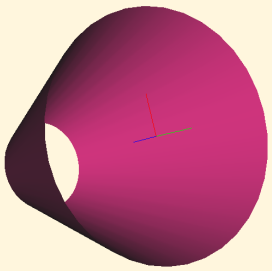
A disk is created as a cone, but with zero length. (This way the base and top radius of the cone become the inner and outer radius of the disk.)



inner, outer radius	The start and end radii (inner, outer) within the local xy-plane.
angle range start, sweep	The start and sweep angle around the rotation z-axis.

4.2.11. Cone

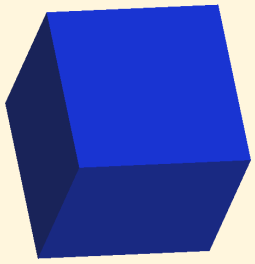
This option creates a cone surface within the given axis system. The cone is defined by a pair of base and top radii and a pair of base and top heights. The cone is extruded into the z-direction of the axis system. The cone may be restricted by a start angle and a sweep angle in the local xy-plane. All angles are defined in degrees and are measured relative to the local x-axis.



base and top cone radius	Defines the base and the top radius of the cone respectively.
length from, to	The start and end position of the cone along the local z-axis.
angle range start, sweep	The start and sweep angle around the rotation z-axis.

4.2.12. Box

Builds a box with the given axis system as local axis system. The size is set by the given xyz-range.



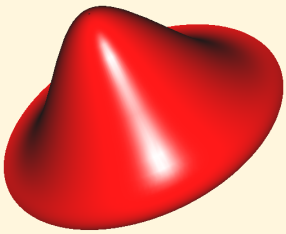
min x,y,z, max x,y,z	The box's xyz-range.

4.3. Procedural Surfaces

This section describes the commands that are available under the menu entry *Procedural Geometry*, offering the user a number of tools to construct surfaces from given curves.

4.3.1. Rotational Surface

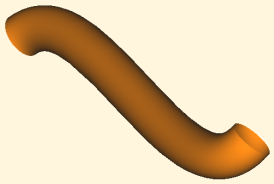
Creates a general rotational symmetric surface. The given axis system is the local coordinate system where the profile curve rotates around the z-axis. The surface may be restricted by a start and a sweep angle in the local xy-plane.



rotation profile curve	The curve to be rotated.
angle range start, sweep	Start angle and sweep angle for the rotation.
ellipse factor	The ellipse factor, i.e. the y/x ratio of the diameters.
local axis system	The local coordinate system of the RotSurface. Refer to description under Section 4.10.1, “Create Axis System ”.

4.3.2. Pipe Surface

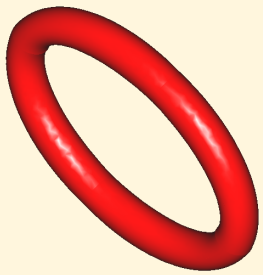
Creates a pipe surface. A pipe surface may be used in fiber optics. It is a 2D circle swept along a spine curve.



profile curve	The curve which should be used to create the pipe.
radius	The radius of the pipe.

4.3.3. Torus

This dialog creates a torus surface within the xy-plane of the given axis system. The size is determined by the profile radius and a pair of radii. The torus may be restricted by a start and sweep polar (θ) and azimuth (ϕ) angles (all angles are defined in degrees).



profile radius	Determines the radius of the torus body.
ellipse radii x,y	Determines the two radii of the ellipse base area, i.e. the base curve of the torus.
start, sweep angle	Start angle and sweep angle to create only a segment of the ellipse.

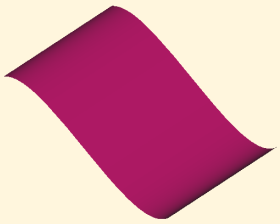
4.3.4. Coil

This dialog creates a coil surface along the z-axis of the given axis system. The start and length/sweep is determined by the creation parameters, the form by a pair of radii and the closeness of the sinuosity by the slope and radius.

radio button: creation type	The creation of the coil can be specified either via two angles (start/sweep) or with a start angle plus length.
start, sweep angle	Determines the start and sweep angle of the coil. (Together with the slope, this determines the dimension of the coil.)
start angle, length	Determines the start angle and length of the coil. (Together with the slope, this determines the dimension of the coil.)
radius in u,v	Determines the two radii of the ellipse base area in u and v.
slope	Determines the ascent of the coil along the given axis.
radius	Determines the radius of the coil body.

4.3.5. Extruded Surface

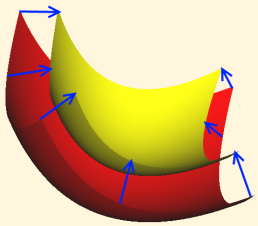
Creates an extruded surface. The surface is created by extruding a curve along a given direction.



profile curve	The curve which should be extruded.
extrusion direction x,y,z	The direction vector.
start length, end length	The start and end of the extruded surface along the extrusion vector.

4.3.6. Offset Surface

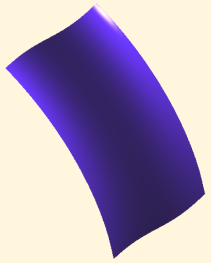
Creates an offset surface. An offset surface is a surface with a given distance from a base surface. The distance is measured in normal direction of the base surface.



base surface	The surface to be offset.
offset	The distance between the base surface and the resulting offset surface.

4.3.7. Swept Surface

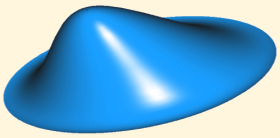
Creates a swept surface. The surface is created by a curve swept along a spine curve.



profile curve	The curve which should be swept along the spine curve.
spine curve	The curve which should be used as spine curve.

4.3.8. Swung Surface

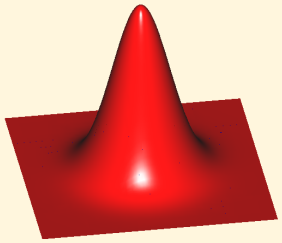
Creates a swung surface. A SwungSurface is a surface like a RotSurface but with a varying radius. The surface is obtained by swinging the 2D profile curve around the Z axis and simultaneously scaling accordingly to the spine curve and by an arbitrary scaling factor alpha. Like in Rot-Surface, the spine and the profile curve should be 2D (X,Y) curves. The z-coordinates are ignored.



profile curve	The curve which should be rotated.
spine curve	The curve which is used to vary the radius.
alpha	An extra scaling factor used for the radius calculation.

4.3.9. Surface from Formula

Creates a surface from an entered formula. The surface may be created as a value grid or as a fitting surface with approximation/interpolation methods.



explicit/parameter form	The formula is either displayed in explicit or in parameter form.
parameter	The parameters of the surface.
formula z = f(x,y)	The explicit formula of the surface.

formula x(u,v), y(u,v), z(u,v)	The formula in parameter form.
# of points X,Y or U,V	The number of points in X,Y or U,V direction, respectively.
X range min, max, Y range min, max	The min, max range of the surface on the X,Y - axis.
U range min, max, V range min, max	The min, max range of the surface in U,V - direction (parameter form).
# patches in U,V (approximation)	The number of patches in U,V.
degree in U,V (approximation, interpolation)	The polynomial surface degree.
U,V smoothness (approximation)	Flag which determines the type of smoothing: point, tangent or curvature continuous.

For a sketch of the dialog, see Figure 4.5, “The Dialog 'Surface from Formula' with sub-dialogs”

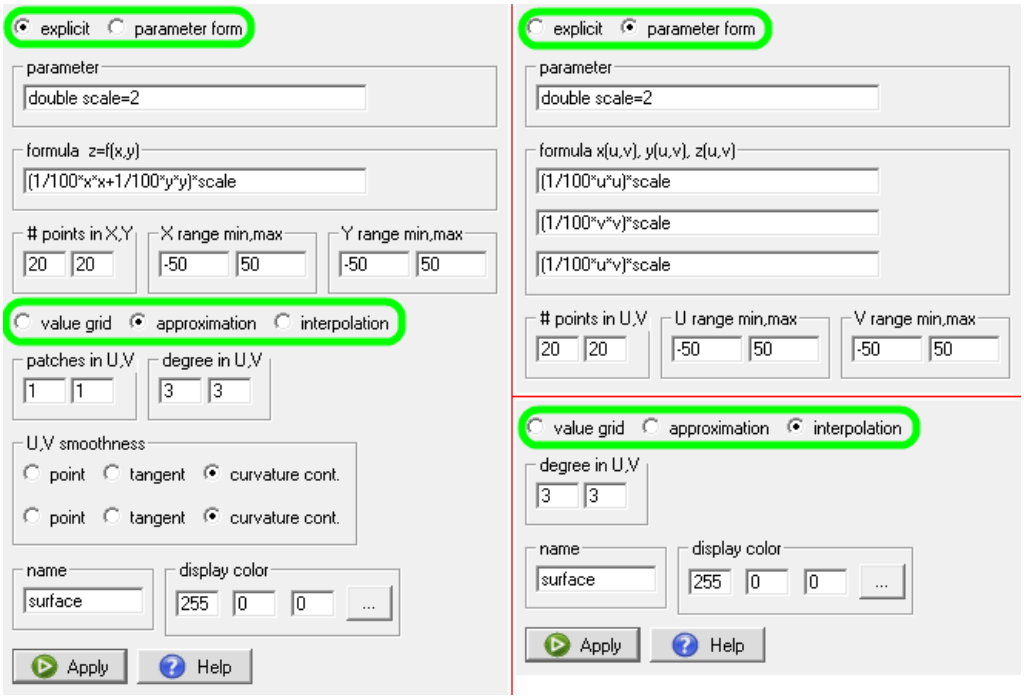


Figure 4.5: The Dialog 'Surface from Formula' with sub-dialogs

4.4. Standard Curves

This section describes the commands that are available under the menu entry *Curves*, offering the user a number of basic curve creation tools.

Most commands start a dialog which merely consists of three parts:

1. A dialog part containing entries specific for the geometry to be created, like the focal length for a parabola or the radius for a circular curve. These parameters are described in the table below.

2. A dialog part controlling the curve's identifier, its coloring and its local axis system. See Figure 4.6, “The parabola dialog with its two parts: [1] geometric specific entries, [2] non-geometric entries (common to most of the dialogs in this section).” for an example.

The second dialog part is the same for most commands.

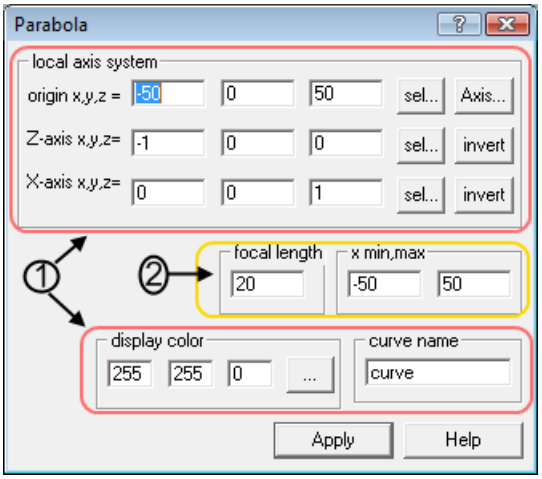
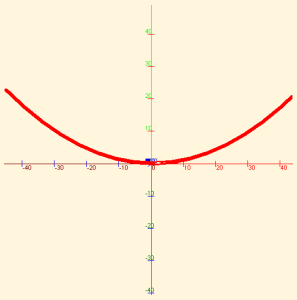


Figure 4.6: The parabola dialog with its two parts: [1] geometric specific entries, [2] non-geometric entries (common to most of the dialogs in this section).

4.4.1. Parabola

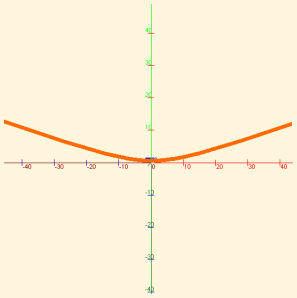
Creates a 3D Parabola. The 3D curve lies in the XY plane of the given axis system and has it's vertex at the axis origin. The focal length and the range define the parabola's shape.



focal length	The focal length of the parabola.
x min,max	The parabola's range in X.

4.4.2. Hyperbola

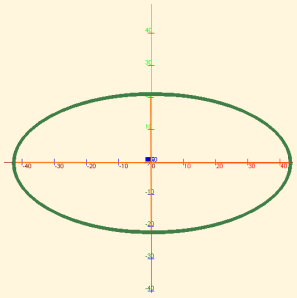
Creates a 3D hyperbola. It lies in the XY plane of the given axis system and is centered at the axis origin. Two focal points define the hyperbola's shape.



f1, f2	The two focal lengths of the hyperbola.
x min, max	The hyperbola's range in x.

4.4.3. Ellipse

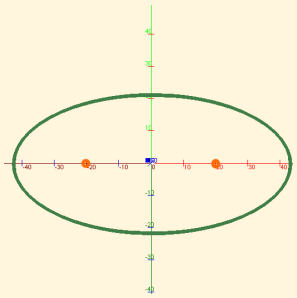
Creates a 3D ellipse. It lies in the XY plane of the given axis system and is centered at the axis origin. The ellipse size is set by 2 radii. A sub arc may be defined by an optional start and sweep angle.



radius u,v	The ellipse radii.
start, sweep angle	The start and sweep angles.

4.4.4. Ellipse by Two Focal Points

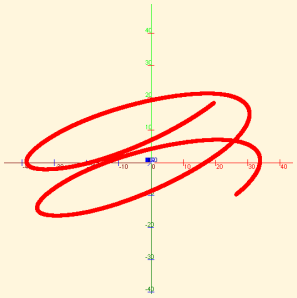
Creates a 3D ellipse. It lies in the XY plane of the given axis system and is centered at the axis origin. The ellipse is set by 2 focal points. A sub arc may be defined by an optional start and sweep angle.



f1, f2	The two focal points of the ellipse.
start, sweep angle	The start and sweep angles.

4.4.5. Helix

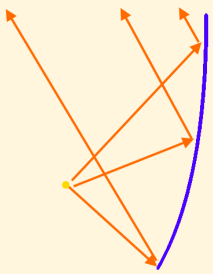
With this tool a curve in form of a helix can be created. It may be used e.g. to create filament shapes in lamp models.



radius u,v	The two radii of the helix. If u=v, the helix has the form of a circle, otherwise it becomes elliptical.
start, sweep angle	The helix is created from start to sweep, where 360/(sweep angle - start angle) defines the number of loops.
slope	This value defines the increase for one loop. The complete length of the helix can thus calculated as follows: slope*360/(sweep angle - start angle).

4.4.6. Compound Parabolic Concentrator

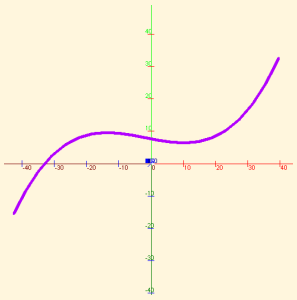
A parabolic arc that sends light coming from a point light source in the arc's focal point in a unique direction.



entry radius	The entry radius of the CPC curve, i.e. the inner radius.
exit angle	The exit angle for the light rays.

4.4.7. Curve from Formula

This tool creates a 2D - curve from an entered formula. The formula may be entered in an explicit form y=f(x) or in a parameter form x(u), y(u), z(u). The curve can be created as a polyline or as a fitting curve with approximation and interpolation methods alternatively.



parameter	The parameters of the curve.
explicit/parameter form	One can either use the explicit or the parameter form of the curve.
formula y(x)	The formula of the curve in explicit form.
formula x(u), y(u), z(u)	The formula of the curve in parameter form.
# of points	The number of points either for the polyline or the fitting curve.
X range min, max	The min, max range of the curve on the X - axis.
polyline /approximation /interpolation	Radio buttons to check which method should be used. The methods are either a polyline or a fitting curve with approximation/interpolation methods.
name	The name of the curve.
degree	The polynomial curve degree.
arcs	The number of arcs to be calculated, used only in the approximation.
smoothing factor	A factor controlling an extra global smoothing. Used only for the approximation method.
smoothness	A selector for the smoothing flag: point, tangent, curvature. Used only for the approximation method.

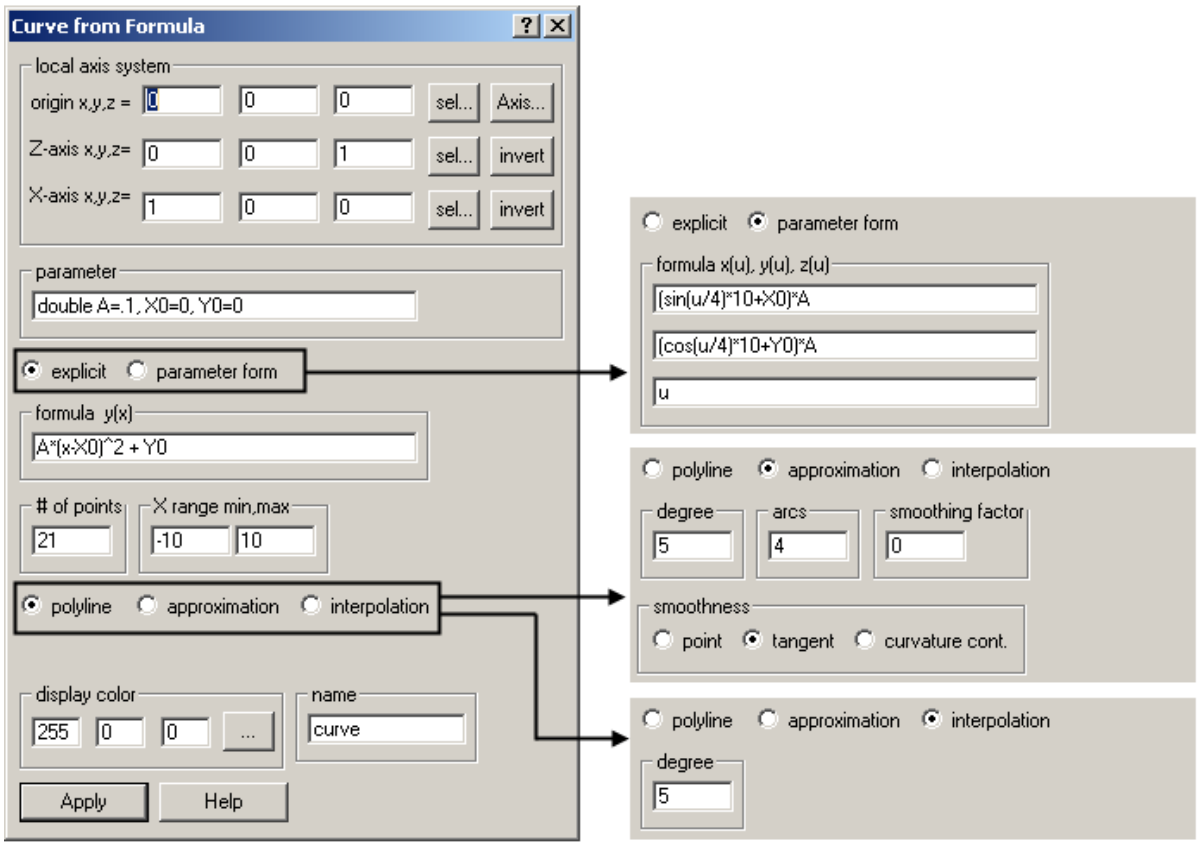


Figure 4.7: The Curve from Formula dialog

4.4.8. Curve between two Curves

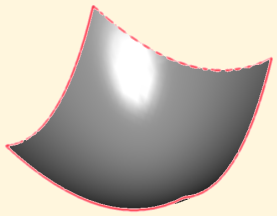
Creates a new surface spanned by two curves

first curve, second curve	The two curves which will be used as borders.

4.4.9. Curves from NURBS Surface Border

Computes the boundary curves from a set of user selected NURBS surfaces and returns them as NURBS curves into the model. The degrees of the computed curves corresponds to the degree of the input surfaces.

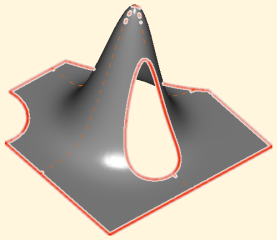
A (non-trimmed) surface's boundary curve is defined as the surface's subsets related to the uv-parameters $u=u_{min}$, $u=u_{max}$, $v=v_{min}$, $v=v_{max}$, where (u_{min},u_{max}) and (v_{min},v_{max}) are the uv-parameter limits.



4.4.10. Curves from Trimmed Surface Border

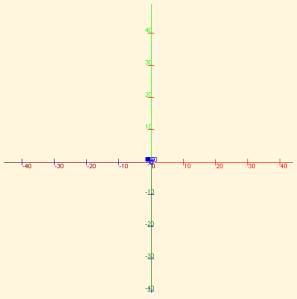
Computes the boundary curves from a set of user selected trimmed surfaces and returns them as NURBS curves into the model. The degrees of the computed curves is fixed to 5.

A trimmed surface's boundary curve is defined as the image of the surface's uv trim bound under the surface $(u,v) \rightarrow (x,y,z)$ mapping.



4.4.11. Create Point

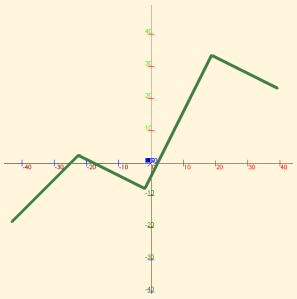
Creates a single point. It can be selected e.g. to define the origin of an axis system or to create a point cloud.



Point x,y,z	The x-,y-,z-coordinate of the point.
point name	The point's name.

4.4.12. Create Polyline

Creates a polyline through a number of points. Existing points can be moved or deleted, also new ones can be added. This does not work like a drawing program (drawing while keep pressed) but via help points (point and click).



line type / line width	Sets the type of the line and the width of selected type.
Interactive Mouse editing	• *View:* Just view the polyline without interaction. • *Sketch:* Move around the mouse and click for adding a new point, while click and hold down the point is moveable. • *Select:* One may select one or more points. • *Insert:* Adds a new point to a selectable section of the polyline (it splits the old section into two new with the new point as a common point). • *Move:* Pick up a single point and change its coordinates.
close/open	Checking the flag *close* tries to create a closure between both the beginning and the end of the polyline (not possible in the sketch mode).
grid mode, grid size	En-/disables the grid-mode; if disabled, one can use freehand drawing without restrictions. If enabled, one can define the solution of the drawing grid via size and then the points of the polyline will automatically be fitted to that grid.
button: snap all to grid	The coordinates of all points are approximated to grid with the given grid size.
view points	This shows all points with their coordinates (view mode only).

The second tab *edit points* allows to determine the single points by their three coordinate values. A point can be inserted either before another point via *insert before* or after the last one via *append*. (*Append* also is used to create the very first point.) An existing point can be erased by pressing *X*.

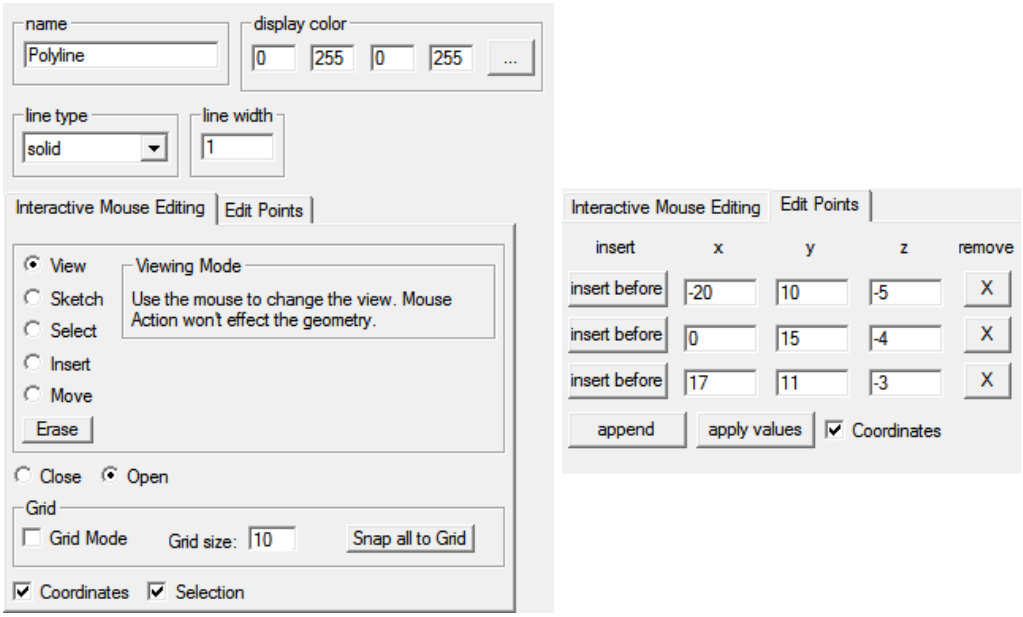
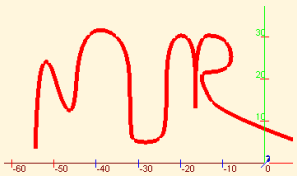


Figure 4.8: The Create Polyline dialog

4.4.13. Create NURBS

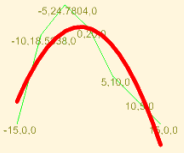
NURBS are a bit of similar to polylines, but with more control of the characteristics of it. As a result of that, the dialog to create NURBS is nearly the same as for Section 4.4.12, “Create Polyline”.



degree	Determines the degree i.e. the order of the polynomial.
knot count	Counts the number of knots of the NURBS (index will be refreshed after pressing *Apply*).
interactive mouse editing / view control points / view knots	For modifying and viewing the control points (see above).
grid mode, grid size	En-/disables the grid-mode; with it disabled one can use freehand drawing without restrictions. If enabled one can define the solution of the drawing grid via size so the points of the NURBS will then automatically be fitted to that grid.
button: snap all to grid	The coordinates of all points are approximated to match the given grid size.

4.4.14. Create Control Curve

Create a Control Curve is a tool which can be used for drawing a curve via parameters and mouse, options are e.g. the polynomial degree and the fitting type.



line type / line width	Sets the type of the line and the width of selected type.
fitting type	Sets the fitting type to approximated, unused or interpolated.
polynomial degree	Determines the degree of the polynomial.
smoothing factor	Determines the used smoothing factor.
interactive mouse editing: viewing mode	Offers the possibilities for editing and viewing the control curve.
grid mode, grid size	En-/disables the grid mode; with it disabled one can use freehand drawing without restrictions. If it is enabled one can define the solution of the drawing grid via size and then the points of the control curve will automatically be fitted to that grid.
button: snap all to grid	The coordinates of all points are approximated to the grid with the given grid size.
values: u-range (min, max)	Sets the min and max for the range in u-direction.
values: list of values in v-direction	One may alter the control points in v-direction.
CSV-file: u-range (min, max)	Sets the min and max for the range in u-direction.
CSV-file: csv file	Selects a control curve file with the specification of the decimal mark.
re-frame	Performs a redraw of the current settings.

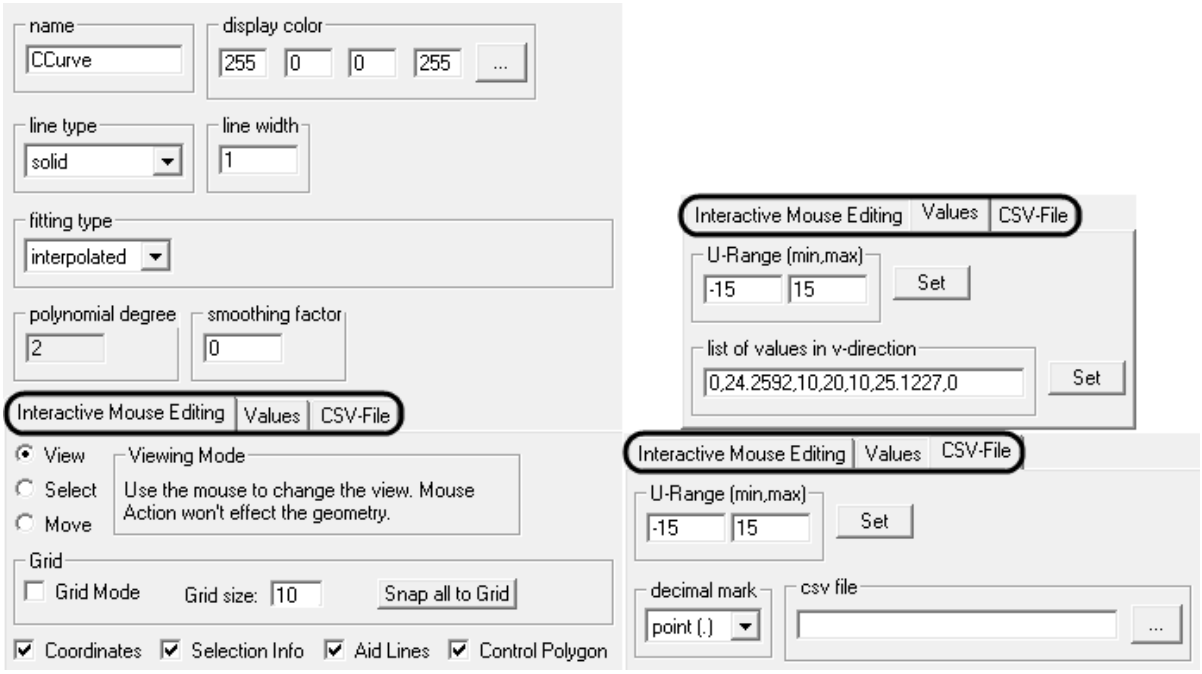


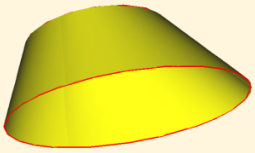
Figure 4.9: The Create Control Curve dialog

4.5. Standard Surfaces

This section describes the commands that are available under the menu entry Create Surface, offering the user a number of basic surface creation tools. The user will be able to create surfaces from already existing curves, points or other surfaces.

4.5.1. Surface from Boundary Layers

Creates a new object using already existing curves as boundary layers.

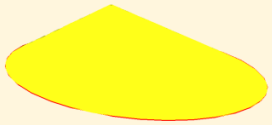


top and bottom curve	These two curves are the base objects spanning the new surface.
left and right curve	These two curves may optionally be chosen to specify the new surface.
projection curve	This curve enables you to project the new surface onto an existing curve.
degree in u, v	Determines the order of the polynomial used to destine the surface.
# patches in u, v	Determines the number of patches used to destine the surface.

deep drawing	The flag deep-drawing enables you to specify the vertex z and the wall height (running from 0 to 1).

4.5.2. Surface between Curve and Point

Creates a new object using an already existing curve and a single point.



rim curve	Determines the curve which will be used.
point x, y, z	Determines the point used as the cone point of the new surface.

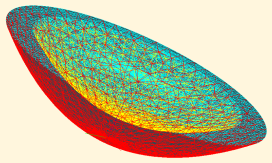
4.5.3. Surface between two Curves

Creates a new surface using two curves spanning it.

first and second curve	Determines the two curves which will used to create the new object.

4.5.4. Surface between two Surfaces

Creates a new surface using two already existing surfaces (the red and yellow ones are the already existing surfaces, the blue one is the new one).



first and second interface	Determines the two surfaces which will used to create the new object.

4.6. Lenses

This section describes various tools for lens creation. Common to all dialogs in this section are the following parameter blocks (which will not be listed again in the subsequent dialog descriptions. Please also see the figure just below.

display color	The display color for the aspherical lens.
lens name	The object's name.
local axis system	The local coordinate system for the lens. See Section 4.10.1, “Create Axis System ”.
material properties	The material properties of the lens. Possible values are • *fresnel:* Switch to Fresnel refraction. This switch is not available for all lens types. • *refraction index (lens medium, outer medium):* The refraction index for the lens medium and the outer medium. • *absorption coefficient (lens medium, outer medium):* The absorption coefficient for the lens medium and the outer medium.

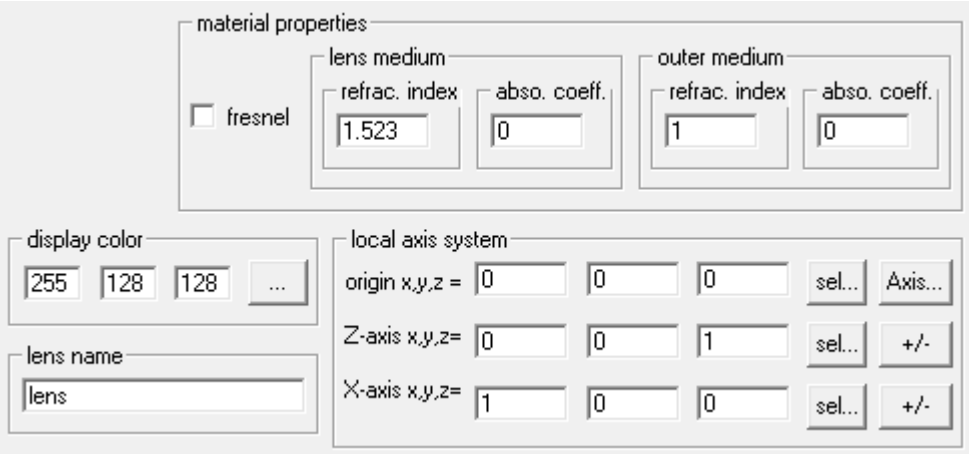
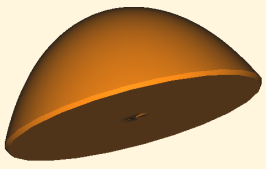


Figure 4.10: The dialog parts common to all lens dialogs in this section

4.6.1. Aspheric Lens

Creates an aspherical lens with the optic either on outer or inner side. The given axis system is the local coordinate system for the lens. The lens is created as a NURBS surface. The lens may be created as a rotational lens with an inner and outer diameter and a start and sweep angle, as an extruded lens with a given width which runs from start width to end width and a length from start length to end length, as an rotational extruded lens with an inner and outer diameter and a start and end length or as a torus lens with a given width, a lower and upper radius and a start and sweep angle. The output angle defines the angle of the outgoing ray. The refraction indexes and absorption coefficients for the lens medium and the outer medium may be set independently.



An aspherical lens has a curvature that changes continuously in the direction of the lens diameter and therefore can eliminate spherical aberration.

polynomial degree	The polynomial degree of the profile curve.
fix side surface	If checked, creates an additional inner surface.
lens type	The lens type. Possible values are: rotational, extruded, rotational extruded, torus.
rotational/extruded/rot. extruded/torus	The lens type.
inner, outer diameter	The two radii of the rotational lens. Usually the inner radius is zero.
start, sweep angle	The start and sweep angle for the rotational surface. The lens is created from start to start + sweep.
thickness, height	The thickness of the lens and the position of the lens on the z-axis.
inner/outer output angle	The outgoing ray angles for the inner and the outer part of the lens.

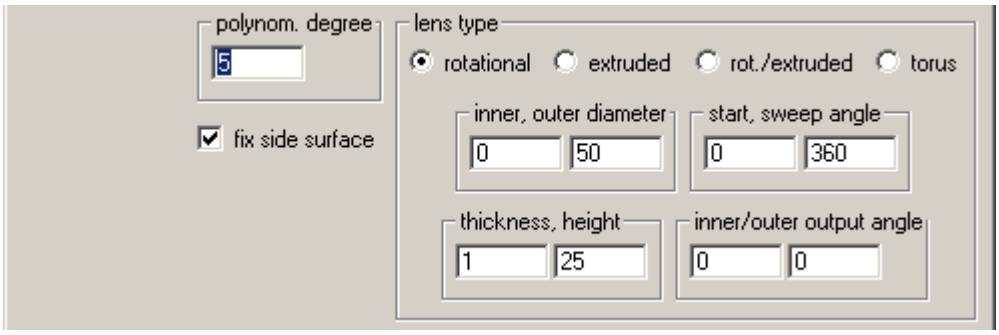
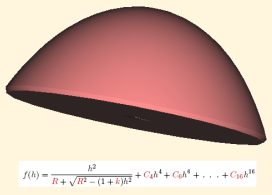


Figure 4.11: The basic aspherical lens dialog (without the set color, set name, axis system and material properties part)

4.6.2. Aspheric Lens Per Formula

Creates an aspherical lens from the given formula, which might be outlined in different ways in different text books. The parameters can easily be entered with the subsequent dialog box.



vertex radius (R)	The vertex radius or radius of curvature alters the curvature of the lens.
conic constant (k)	The conical constant, which alters the shape of the lens. k ‹ -1 hyperboloid; k = -1 paraboloid; -1 ‹ k ‹ 0 ellipsoid; k = 0 circle; k › 0 ellipsoid
coefficients C4, ... , C16	Coefficients for the aspherical formula. By default, these values are set to 0.
lens radius	The radius determines the base size of the lens.
add closing disc	Enables/disables the creation of a closure of the lens.
# discretization points	The number of discretization points for the lens curve approximation.
# arcs	The number of arcs for the lens geometry.
polynomial degree	The polynomial degree of the profile curve; recommended: 5.

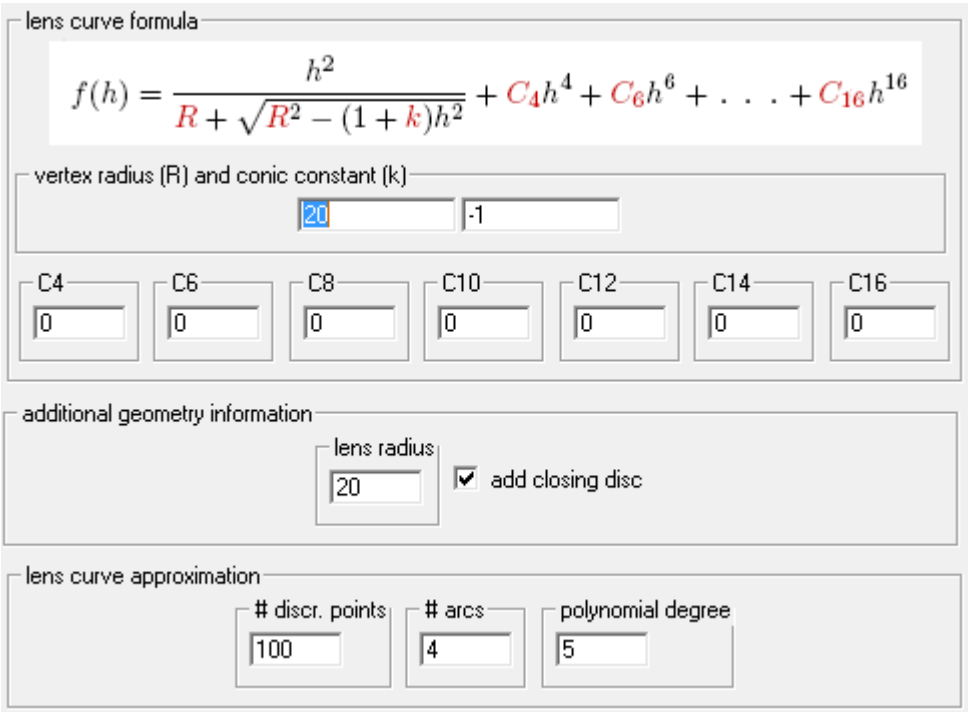
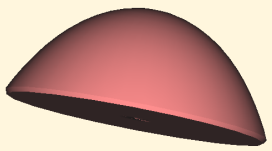


Figure 4.12: The Aspherical Lens Per Formula dialog

4.6.3. Free Form Fresnel Lens

Creates a Fresnel lens with "ideal" free form curved rings or flat elements alternatively. The shape consists of a flat cone surface on one side and a rotational surface with a free form refraction curve ring on the other side. The refraction index and absorption coefficient may be set differently for the inner (lens) and outer medium. The Fresnel flag of the Refractor object may be set (0 or 1). The center and the outer rim of the lens can also be on different heights. This way the flat side may be a cone instead of a flat round disk. If the inner diameter is greater zero, the lens has an inner



hole. In this case, a "hole" surface is created as well. The lens is divided into N equal sized rings. The lens focal point lies in the origin of the given axis system. The optical direction is the z-axis. The lens may be created as an rotational lens, an extruded lens, an extruded symmetrical lens, a rotational and extruded lens, a torus lens or as a torus symmetric lens with different parameter to be entered. with different parameter to be entered.

lens type	One can create one of the following lenses: rotational, extruded, extruded symmetric, rotational extruded, torus and torus symmetric.
base line / curve	For the creation of the lens a predefined base line may be used or otherwise an external curve be selected. In case of base line the inner, outer height is used.
inner, outer diameter	The two radii of the lens. Normally, the inner radius is 0.
inner, outer height	The two heights of the flat side. Normally, both sides have the same height and the flat side becomes a disk.
start angle, sweep angle	The start and sweep angle for the rotational surface. The lens is created from start to start + sweep. Also used for torus and torus symmetrical lend.
start, end width	The start and end width of the extruded lens. The lens is created from start to start + end.
start, end length	The start and end length of the extruded lens. The lens is created from start to start + end.
width	The extension of the extruded symmetric lens in x-direction.
start, end ring height	The start and the end z-position of the torus lens.
inner, outer ring radius	The inner and the outer ring radius of the torus and torus symmetric lens.
focal radius	The focal radius of the torus and symmetric torus lens. The total radius of the toroidal lens is the inner or outer radius plus the focus radius.
ring height	The height of the torus symmetric lens.
flat prism	Check to create flat prisms instead of profiled curves for the elements. A flat prism has the polynomial degree of 1.
fix side surface	Check to create a bottom side with the entered thickness.
profile delta R	The gap delta between the profile arcs.
number of rings	The number of rings.
thickness	The wall thickness between flat side and optic.
polynomial degree	The polynomial degree of the lens.
light target	The target type. • *Far field:* the outgoing rays leave the lens towards the direction of the given target angles.

	• *Near field:* the outgoing rays leave the lens toward the entered position on the target z-plane. • *Virtual near field:* the rays leave the lens as if coming from a start point on the virtual z-plane.
target angles	The ray output angles (if a far field is used (any number)).
target R positions	The desired ray output position radii on the target z-plane (if a near field is used; any number).
target z-plane	The position of the target plane in z-direction.
virtual R positions	The ray output position radii on the virtual z-plane (if a virtual near field is used; any number).
virtual z-plane	The position in z-direction of the virtual z-plane.
fresnel	Switch to Fresnel refraction.

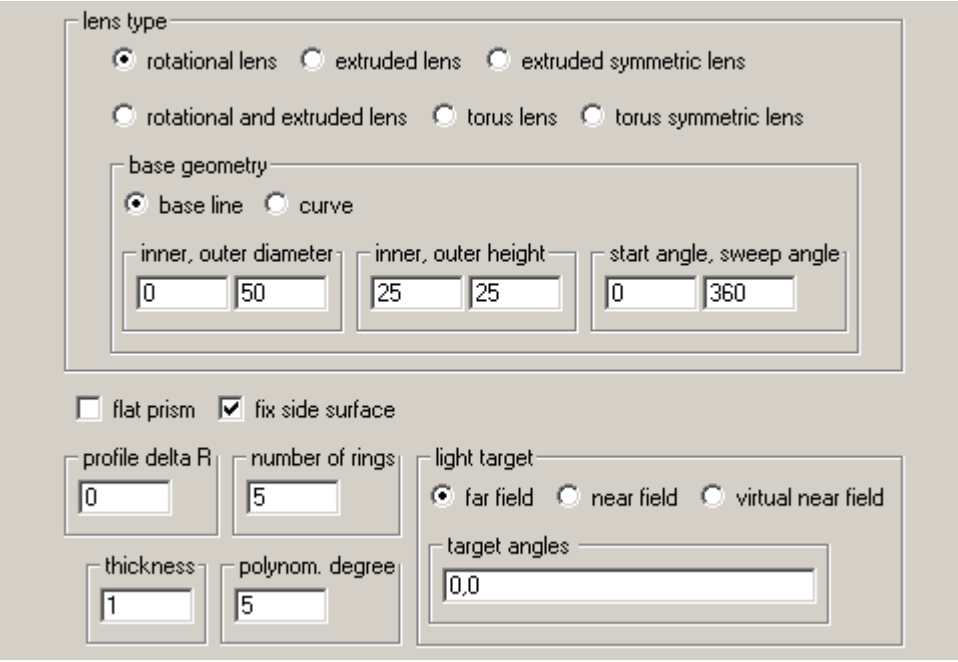
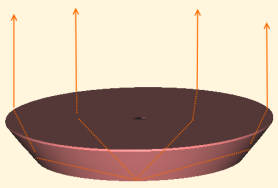


Figure 4.13: The FF Fresnel Lens dialog

4.6.4. TIR (Total Internal Reflection) Lens

Creates a TIR (Total Internal Reflection) lens, particularly suitable for LED applications. A TIR lens parallelizes light from lambertian sources by making use of the lens' internal reflections.



The first options are identical to those of the previous lens (Free Form Fresnel Lens).

# rings	Defines the number of rings of the lens.
fix side surface	This flag enables/disables a closing surface on the exit face.
thickness	Defines the thickness of the lens.
light target	The target type. • *Far field:* the outgoing rays leave the lens towards the direction of the given target angles. • *Near field:* the outgoing rays leave the lens toward the entered position on the target z-plane. • *Virtual near field:* The rays leave the lens as if coming from a start point on the virtual z-plane.
target angles	The ray output angles (if a far field is used (any number)).
target R positions	The desired ray output position radii on the target z-plane (if a near field is used; any number).
target z-plane	The position of the target plane in z-direction.
virtual R positions	The ray output position radii on the virtual z-plane (if a virtual near field is used; any number).
virtual z-plane	The position in z-direction of the virtual z-plane.
fresnel	Switch to Fresnel refraction.

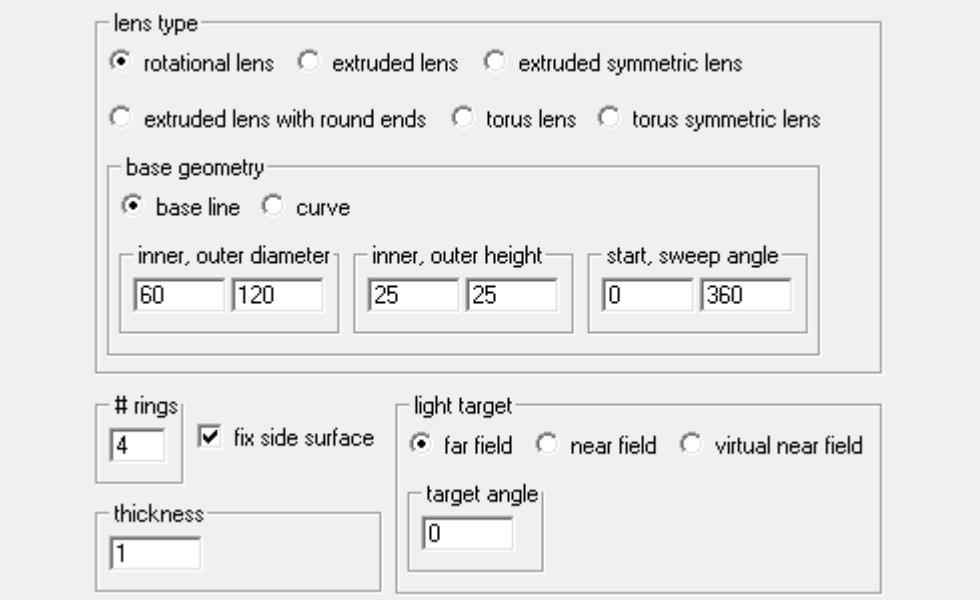
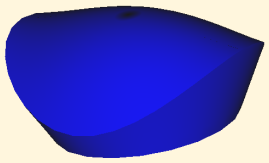


Figure 4.14: The TIR Lens dialog

4.6.5. Potato Chip Lens

Creates a multi surface shape object for a variable aspherical lens in potato chip form. The shape consists of a cone surface on one side and an optical surface with four FF refraction curves on the other side. These four curves are called the "west", "north", "east", and "south" curve. For each curve, lists with 4 value pairs with factorX and factorY in each pair control the creation of each patch (NW,NE,SE,SW) in the VaroSurface. The refraction index and absorption coefficient may be set differently for the inner (lens) and outer medium. The Fresnel flag for the refractor object may be set (0 or 1). For each curve N,S, E, and W the output angle may be entered.



polynomial degree	The polynomial degree of the profile curve.
inner, outer diameter	The two radii of the lens. Normally, the inner radius is 0.
inner, outer height	The two heights of the flat side. Normally, both sides have the same height and the flat side becomes a disk.
thickness	The thickness of the lens, i.e. the thickness in z-direction.
fix side surface	Enables/disables the creation of a fix side of the lens.
output angle N, S, E, W	The ray output angle for the four different curves.

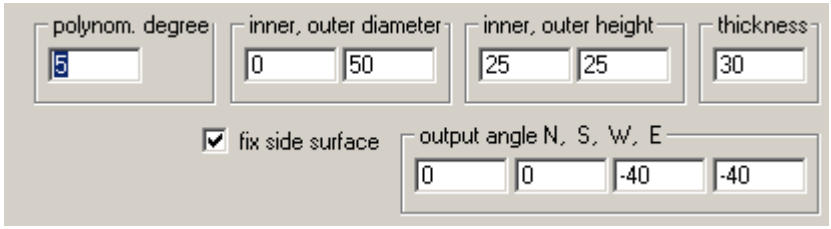
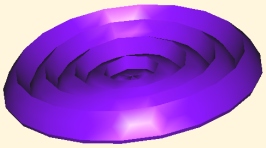


Figure 4.15: The Potato Chip Lens dialog

4.6.6. Free Form Variable Fresnel Lens

Creates a variable Free Form Fresnel lens. The dialog box and nearly all input parameter are described in FF Fresnel Lens {Outer Optic, Inner Optic}. The additional parameter "weight for VaroSurface" and additional "light target" parameter are described below.



weight for VaroSurface	The weight for the surfaces of the ff varo Fresnel lens. Values of 0.5 for both, up-down and left-right are leading to a round shape. • *delta up-down:* The weight for the varo surface up/down. • *delta left-right:* The weight for the varo surface left/right.

target angles NS	The ray output angles in NS direction (if a far field is used (any number)).
target angles WE	The ray output angles in WE direction (if a far field is used; any number).
target R positions NS	The desired ray output position radii on the target z-plane in NS direction (if a near field is used; any number).
target R positions WE	The desired ray output position radii on the target z-plane in WE direction (if a near field is used; any number).
target z-plane	The position of the target plane in z-direction.
virtual R positions NS	The ray output position radii on the virtual z-plane in NS direction (if a virtual near field is used; any number).
virtual R positions WE	The ray output position radii on the virtual z-plane in WE direction (if a virtual near field is used; any number).
virtual z-plane	The position of the virtual z-plane in z-direction.

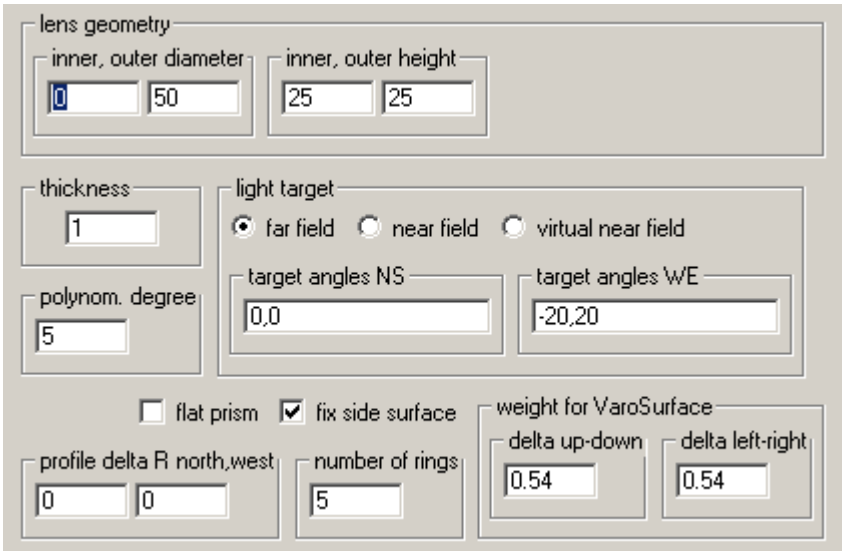
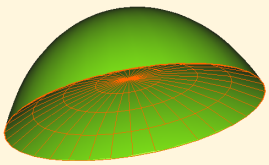


Figure 4.16: The FF Variable Fresnel Lens dialog

4.6.7. Elliptic Lens

Creates an elliptical lens. The lens consists of an elliptical outer side and a spherical inner side. The local axis system is the local coordinate system of the lens. Those rays which start from the center of this axis system, leave the lens parallel.



The lens may either be created as a rotational lens with a given diameter and a start and sweep angle, as an extruded lens with a given width and a start and end length or as a toroid lens with a radius, height and a start and sweep angle.

The available options and parameters depend on the currently selected lens type.

vertex length	The vertex point of the lens on the z-axis of the local coordinate system.
diameter	The diameter of the rotational lens.
start angle, sweep angle	The start and sweep angle of the rotational and toroidal lens.
fix side surface	Enables/disables the creation of a fix side of the lens.
vertex width	The width of the extruded lens.
start length, end length	The start and end length of the extruded lens.
radius, height	The radius and height of the toroidal lens.
focus radius	The focus radius of the toroidal lens.

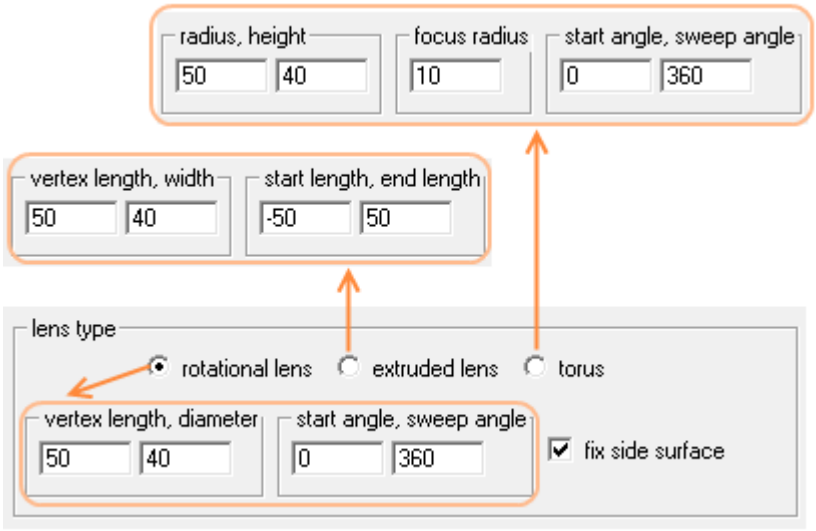


Figure 4.17: The Elliptic Lens dialog

4.6.8. Hyperbolic Lens

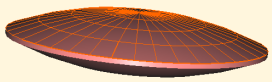
Creates a hyperbolic lens. The lens consists of a hyperbolic inner and a flat outer side. The local axis system is the local coordinate system of the lens. Those rays which start from the center of this axis system, leave the lens parallel.

The lens may either be created as a rotational lens with a given diameter and a start and sweep angle, as an extruded lens with a given and a start and end length or as a toroidal lens with a radius, height and a start and sweep angle.

The dialog parameters for the hyperbolic lens are exactly the same as for the elliptical lens (see Section 4.6.7, “ Elliptic Lens”).

4.6.9. Spherical Lens

Creates a spherical lens. The lens is made up of multiple surfaces. The local axis system is the local coordinate system of the lens. The refraction indexes and absorption coefficients for the lens medium and the outer medium may be set independently. The flag Fresnel considers Fresnel reflection.



The spherical lens comes in four different flavours: Planar Convex, Planar Concave, Biconvex and Biconcave. The two "planar" versions have a planar closing surface, while the "Bi" versions mirror the original (concave or convex) surface in order to obtain the full lens shape.

geometry properties	• *focal length:* The focal length of the lens. • *diameter:* The diameter of the lens. • *thickness:* The thickness of the lens. • *radii ratio:* The ratio of the two different radii for the types Biconcave and Biconvex.

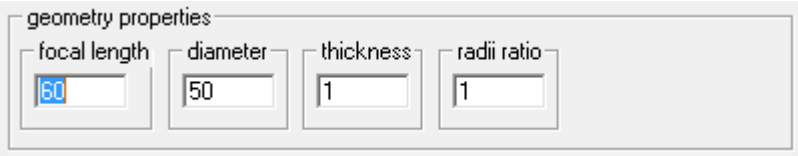


Figure 4.18: The Spheric Lens dialog

4.7. Approximated and Interpolated Curve

This section presents various tools to read point data from external files and generate continuous curves from the data via interpolation or approximation. The dialogs consist merely of three parts:

1. Non-geometric properties like curve name and curve color.
2. Reading the point data from a file. This only requires the input of a path and file name.
3. Setting interpolation/approximation parameters like polynomial degree, number of polynomial arcs etc. See Figure 4.19, “Curve interpolation/approximation parameters”.

The following table describes the interpolation/approximation parameters:

approximation / interpolation	Defines whether the curve will be approximated or interpolated.
degree	The polynomial curve degree.
arcs	The number of arcs to be calculated, used only for the approximation method.
smoothing factor	A factor controlling an extra global smoothing. Used only for the approximation method.
smoothness	A selector for the smoothing flag: point, tangent or curvature. Used only for the approximation method.

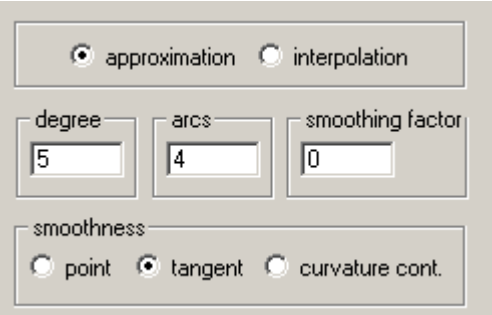
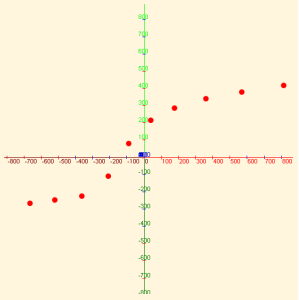


Figure 4.19: Curve interpolation/approximation parameters

4.7.1. Read Point Row

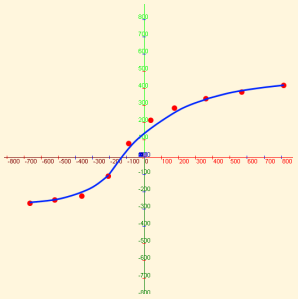
Reads a PointRow data object from a file. The file must have (x,y,z) co-ordinates or (x,y) coordinates (in case of is2D==1) in one line. Between the three (or two) numbers there may be any alphanumeric text which will be skipped when reading the file. Example of a PointRow data file:



is2D	If switched on, the point row lies in the xy-plane, with z=0.
Select point row file	Enter the name of the file which contains the point row data.

4.7.2. Create Fitting Curve

Fits a curve through a PointRow data object, preferably read from a file. The curve may be either approximated or interpolated through the points. In case of interpolation, normals at each point may be used, represented by another 3 numbers on each line.



point row	The point row, from which the fitting curve should be created.
read point row	Reads a point row from a file. See Section 4.7.1, “ Read Point Row”.

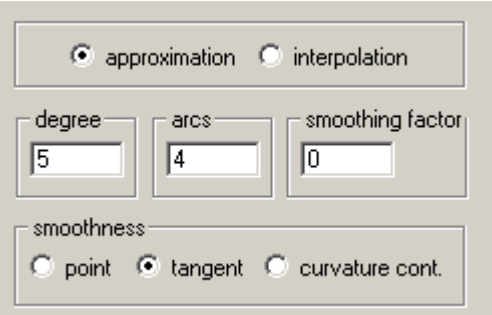
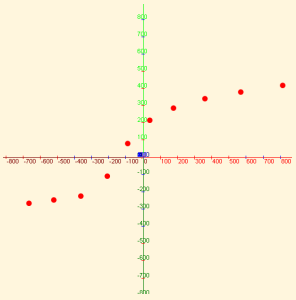


Figure 4.20: The Create Fitting Curve dialog

4.7.3. Read Value Row

Reads a ValueRow data object from a data file, consisting of x,y coordinates in one line, the coordinates separated by a comma. Example for a ValueRow data object:

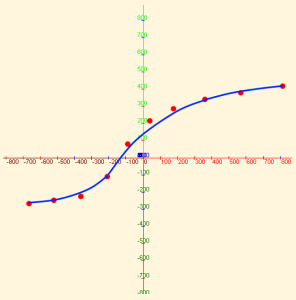


Select value row file	Enter the name of the file which contains the value row data.

4.7.4. Create Explicit Curve

Creates an explicit curve which has the explicit mathematical form: y=f(u) and is stored internally as a NURBS curve. The curve is defined by a ValueRow data object of length nu consisting of a vector with nu U parameters and a vector with nu Y values. The sequence of the U parameters must be in a constantly increasing or decreasing order.

The NURBS data is calculated during the object construction by an interpolation or approximation algorithm.



value row	The value row from which the explicit curve is created.
read value row	Reads a value row from a file, see also Section 4.7.3, “ Read Value Row”.
local axis system	The local axis system of the curve. Refer to the description under Section 4.10.1, “Create Axis System ”.

4.8. Approximated and Interpolated Surface

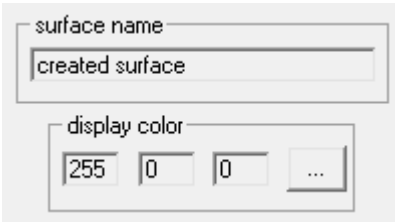
This section presents various tools for point grid and point cloud handling, including surface generation via interpolation or extrapolation.

4.8.1. Common Dialog Parts

Some dialog parts are common to most of the tools described subsequently. Instead of repeating those parts in every subsection, we only describe them in detail here:

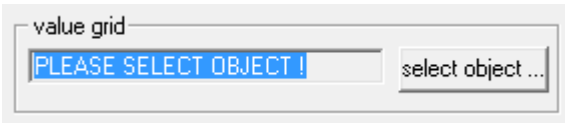
- object's color and name:

display color	The object's color as displayed in the Geo View
name	The object's name



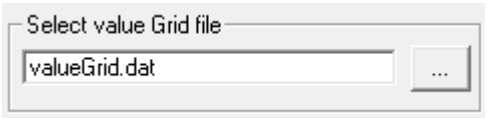
- interactive selection of objects (grids, surfaces):

select object	Enters the object selection mode to select an object from the Geo View or the Tree View.



- import of objects from external files

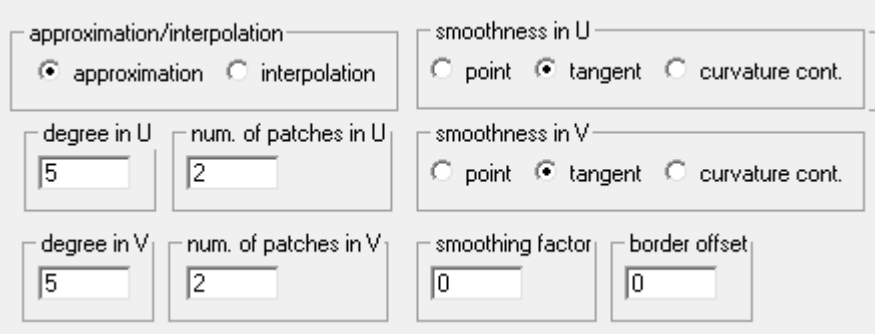
read object	Reads an object from an external file. The file path may be entered via a file selection dialog.



- Parameters for interpolating/approximating a surface from discrete point data:

approximation / interpolation	Selector for the approximation/interpolation method.
degree in u,v	The polynomial surface degree.
number of patches in u,v	The number of NURBS patches to be calculated, used only for the approximation method.

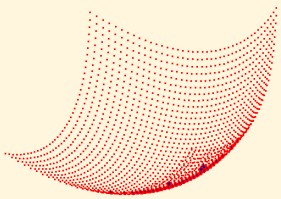
smoothness in u,v	A selector for the smoothing flag: point, tangent or curvature. Used only for the approximation method.
smoothing factor	A factor for controlling an extra global smoothing. Used only in the approximation method.
border offset	A factor for an offset at the border, the final offset is factor * Urange in u-direction and factor * Vrange in v-direction. Used only in the approximation method.



- Axis systems, see Section 4.10.1, “Create Axis System ” for a description.

4.8.2. Read Point Cloud

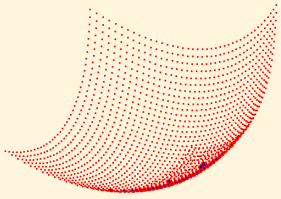
The point cloud data can be read from a data file. For the current implementation the data file must have the x,y,z coordinates in one line. There may be any alphanumeric text between the three numbers. This text will be skipped when reading the file.



Select point cloud file	See Section 4.8.1, “Common Dialog Parts”.

4.8.3. Create Point Cloud

Creates a point cloud via point input, point selection or grid selection. For point input, the x,y,z - coordinates of the points have to be entered. The selection methods offer the possibility to select points or point grids from the tree view.



points x,y,z for Point Cloud	Here the x,y,z - coordinates for each point are entered.
# points	Defines the number of points to be entered. A corresponding number of input fields for the coordinates appear as soon as update is pressed.
grid selection list	The selected point grids are displayed in this list.
update	Updates the dialog according to the number of entered points.
clear list	All entries in the list are deleted.
select...	• *Point selection:* if pressed, points can be selected from the tree view. • *Grid selection:* like above, but point grids have to be selected. In both cases, a list is created which contains the selected points / point grids. This list is used for the point cloud creation.
display color, point cloud name	See Section 4.8.1, “Common Dialog Parts”.

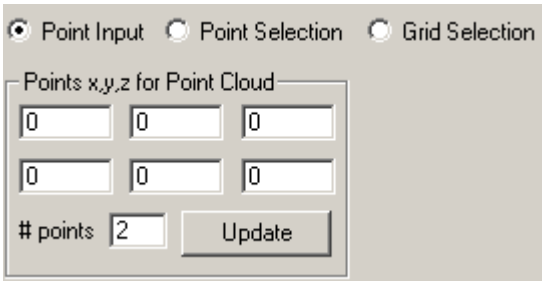
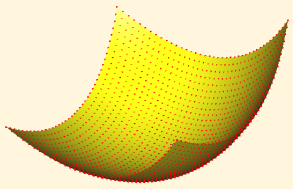


Figure 4.21: The Create Point Cloud dialog

4.8.4. Sample Point Cloud from Target Scene

Creates a new point cloud from a target scene. The tool uses all points in the start grid as start points and shoots rays into the target scene in order to get a new point and normal. The way this is done is controlled by the sample method flag.



use sample surfaces	A selector whether to use all surfaces or only the selected surfaces in the target scene.
grid sample method	A selector for the sample method: • *with direction vector:* All rays are created from each point with a given direction. The rays will intersect with the first surface in the target scene. The intersection points and the normals are stored in the resulting point cloud.

	• *from sphere center:* Each ray starts from a given start center and goes through the points. The rays will intersect with the first surface in the target scene. The intersection points and the normals are stored in the resulting point cloud. • *from triangle vertices:* Creates a point cloud from triangle vertices. • *directionX, directionY, directionZ:* The sample direction vector. • *centerX, centerY, centerZ:* The start point of the rays. The sample direction goes from the start center to the grid point.
create sample point grid	Creates a sample point grid, see also Section 4.8.7, “Create Point Grid ”.
select point grid, read point grid, point cloud name, display color	See Section 4.8.1, “Common Dialog Parts”.

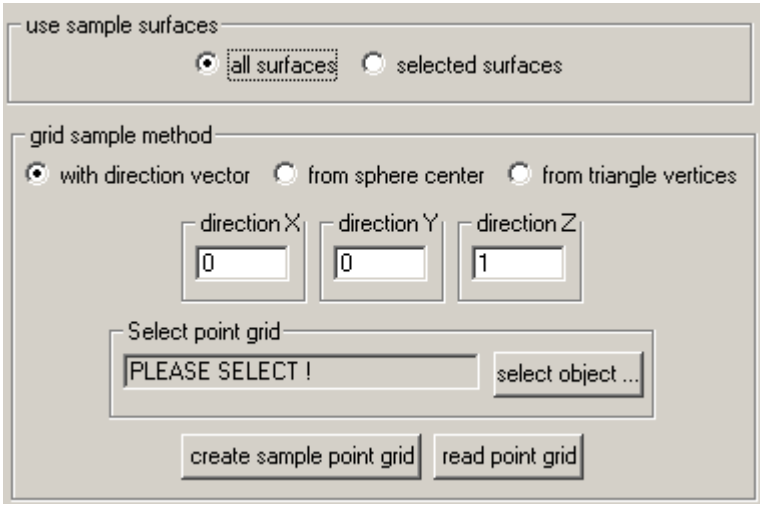
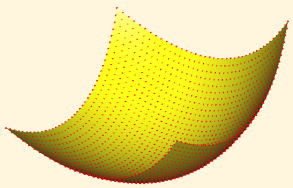


Figure 4.22: The Sample Point Cloud from Target Scene dialog

4.8.5. Create Surface from Point Cloud

Creates a NURBS Surface from a given point cloud. The NURBS data is calculated during the object construction by an interpolation or approximation algorithm.



create uv-parametrization	A flag to create an uv-parametrization. If not selected, the point cloud must have it's own uv-values. The projection can either be accomplished with a plane or a sphere. If spherical projection is chosen, the local axis system can be of type B, A or C.
use smallest uv quad	A flag to use the smallest uv quadrilateral.
select point cloud, read point cloud, result surface parameter, surface name, display color	See Section 4.8.1, “Common Dialog Parts”.

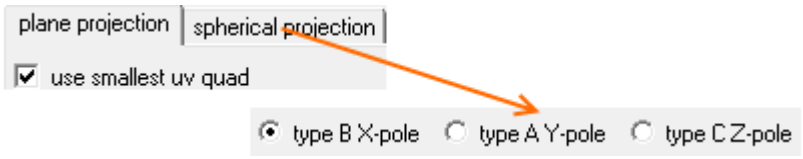
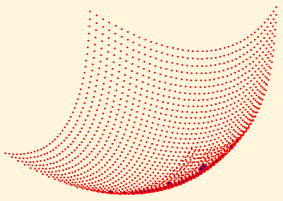


Figure 4.23: Detail of the "Create Surface from Point Cloud" dialog.

4.8.6. Read Point Grid

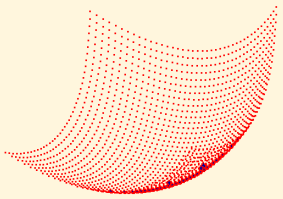
The point grid data can be read from a data file. For the current implementation the data file must have x,y,z coordinates or x,y coordinates (in case of is2D==1) in one line (the same as in point row). Between the three (or two) numbers there may be any alphanumeric text. This text will be skipped when reading the file.



is2D	If checked, the point grid lies in the xy-plane.
nu, nv	The length of the u and v parameter array.
Select point grid file	See Section 4.8.1, “Common Dialog Parts”.

4.8.7. Create Point Grid

Creates a point grid with n by m points in U and V in a patch (p1,p2,p3,p4).



number of points in u,v	The number of points in U and V direction.
range min_U, max_U, min_V, max_V	The parameter range in u- and v-direction.
the four corner points of the patch	The (x,y,z)-coordinates of the four corner points of the patch.
select point grid, display color	See Section 4.8.1, “Common Dialog Parts”.

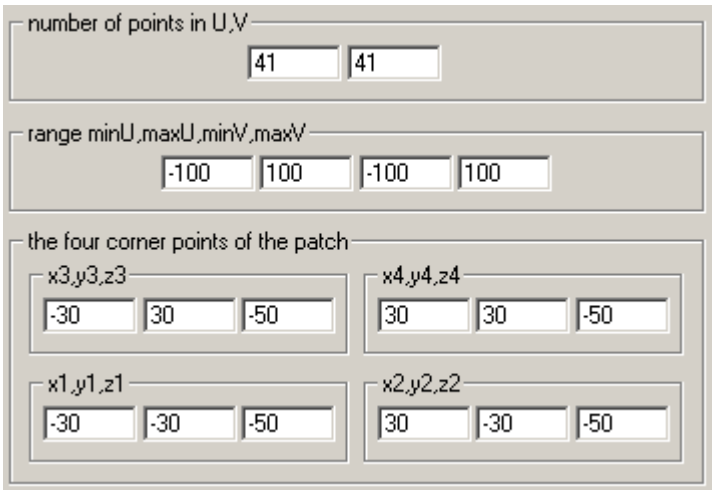
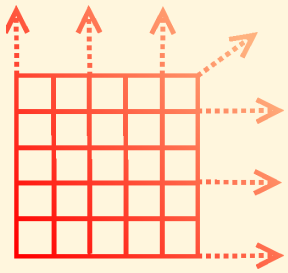


Figure 4.24: The Create Point Grid dialog

4.8.8. Extrapolate Point Grid

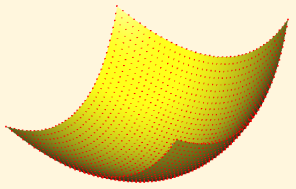
Extrapolates a point grid.



selection point grid	The point grid to be extrapolated.
extrapolation method	Specify here the extrapolation direction. Options are in u-direction, in v-direction or in both directions.

4.8.9. Sample Point Grid from Target Scene

Creates a new point grid by sampling over surfaces. The tool uses all points in the start grid as start points and shoots rays into the target scene in order to get a new point and normal. The way this is done is controlled with the sample method flag.



use sample surfaces	A selector whether to use all surfaces or only the selected surfaces in the target scene.
grid sample method	A selector for the sample method: • *with direction vector:* All rays are created from each point with a given direction. The rays will intersect with the first surface in the target scene. The intersection points and the normals are stored in the result point cloud. • *from sphere center:* Each ray starts from a given start center and goes through the points. The rays will intersect with the first surface in the target scene. The intersection points and the normals are stored in the resulting point cloud. • *directionX, directionY, directionZ:* The sample direction vector. • *centerX, centerY, centerZ:* The start point of the rays. The sample direction goes from the start center to the grid point.
create sample point grid	Creates a sample point grid, see also Section 4.8.7, “Create Point Grid ”.
select point grid, read point grid	See Section 4.8.1, “Common Dialog Parts”.

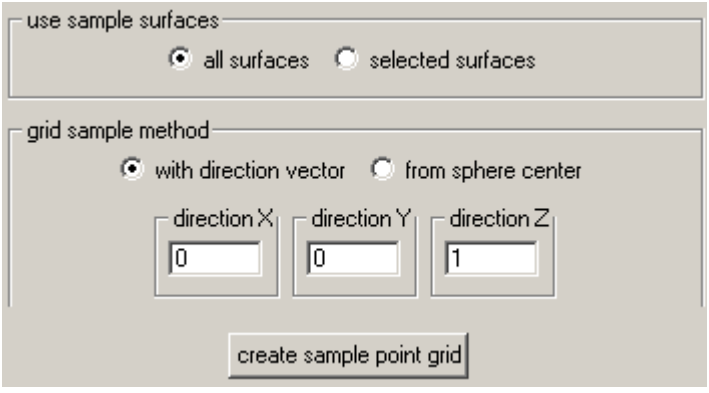
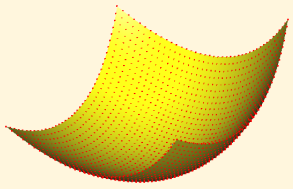


Figure 4.25: The Sample Point Grid from Target Scene dialog

4.8.10. Create Point Grid on Surface

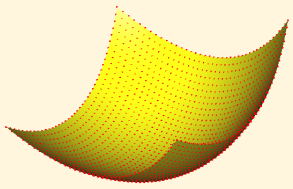
A point grid with n by m points in U and V is created on a surface.



number of grid points in U,V	The number of points in U and V direction.
select base surface, point grid name, display color	See Section 4.8.1, “Common Dialog Parts”.

4.8.11. Create Fitting Surface

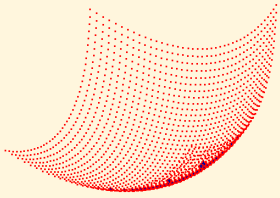
Fits a surface through a regular grid of points. The surface may be either approximated or interpolated through the points. The surface is stored as a NURBS surface. The NURBS data is calculated during the object construction by interpolation or approximation. The surface itself is defined by a point grid of length nu*nv consisting of a vector with nu U parameter, a vector with nv V parameters and three vectors with nu*nv X,Y,Z triples.



select point grid, read point grid, surface approximation parameters, surface name, display color	See Section 4.8.1, “Common Dialog Parts”.

4.8.12. Read Value Grid

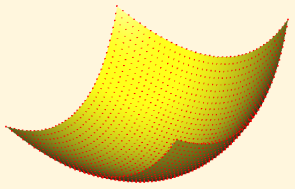
The value grid data can be read from a data file. For the current implementation the data file must have nu*nv values in one column.



nu, nv	The length of the U and V parameter array.
select value grid file	See Section 4.8.1, “Common Dialog Parts”.

4.8.13. Create Explicit Surface

Creates an explicit surface from a value grid. The surface has the explicit mathematical form z = f(x,y) and is stored as a NURBS surface. The NURBS data is calculated during the object construction by an interpolation or an approximation algorithm. The surface is defined by a value grid of length nu*nv. It consists of a vector with nu U parameters, a vector with nv V parameters and a vector with nu*nv Z values. The sequences of the U and V parameters have to be in a constant increasing or decreasing order. The value grid object may have more than one matrices of Z values. The used matrix is chosen by an index.

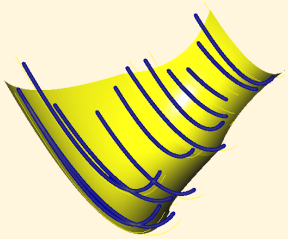


select value grid, read value grid, surface approximation parameters, surface name, display color, local axis system

See Section 4.8.1, “Common Dialog Parts”.

4.8.14. Create Skinned Surface

Creates a surface from a NURBS curve row. A skinned surface is a surface created by an approximation or interpolation through a set of curves.

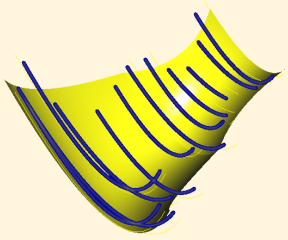


select NURBS curve row, surface approximation parameters, display color, surface name

See Section 4.8.1, “Common Dialog Parts”.

4.8.15. Create Skinned Surface Loft

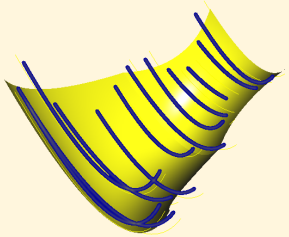
The same as above only that the NURBS curve row is created by single curves which can be selected. Different approximation and interpolation methods are available.



curve selection list	A list which contains the selected curves.
(extra) surface approximation parameters	• *piecewise:* The surface will be piecewise interpolated, where the number of curves has to be N*degree+1. • *tight:* The surface is calculated with cubic spline interpolation. the U parameterization along the interpolation direction orthogonal to the curve is calculated by the average square root of the chord length between the curves. This corresponds to the Rhino3D loft "tight" method. • *normal:* the surface is calculated with cubic spline interpolation. The U parameterization along the interpolation direction orthogonal to the curve is calculated by the average chord length between the curves. This corresponds to the Rhino3D loft "normal" method.
NURBS curve selection, (standard) surface approximation parameters, display color, surface name	See Section 4.8.1, “Common Dialog Parts”.

4.8.16. Create Skinned Surface Horizontal/Vertical

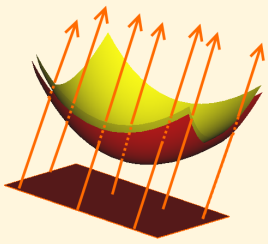
Creates a surface from two NURBS curve rows, the result is projected on a selected surface.



curve selection list	Press *select object ...*, mark the wanted curves in the TreeView, then press *selection done* . This has to be done separately for the horizontal and the vertical list.
projection surface	The resulting surface will be projected on this selected base surface.

4.9. Ray Deviation Correction

Computes a ray deviation correction lens. For a given start surface, a ray direction, a starting point, a starting distance and a refraction index this function computes a "neutral" lens. That is, a second "correction" surface is computed in such a way that light rays with the given direction pass the lens shape (consisting of the base surface, the second surface plus the medium defined via the refraction index) without changing their direction.



base surface	The start surface, from which a second surface should be created.
start u,v	The initial surface parameters specify the starting point of the computation in relative surface coordinates. For example, u=v=0.5 starts the computation in the middle of the base surface.
refraction index n	The refraction index of the medium between the two surfaces.
initial thickness	The initial distance of the two surfaces at the designed u,v surface parameters.
calculation step size	The numerical step size in u and v used for the calculation.
ray direction	The light direction, i.e. the direction of the incoming rays.
check thickness after computation	Switch to enable the creation of a thickness map during the surface calculation. At the designed u,v starting parameters, the thickness should equal the given initial thickness.
name of thickness data	The name of the thickness map.
number of points in u,v	The number of points in u- and u- direction used for the thickness map.
shape name	The name of the shape that represents the lens.
create shape	If checked, creates a lens shape containing the base surface, the computed "correction" surface plus the medium defined via the given refraction index.
result surface parameters	Common surface approximation parameters, see for instance .Section 4.8.1, “Common Dialog Parts”

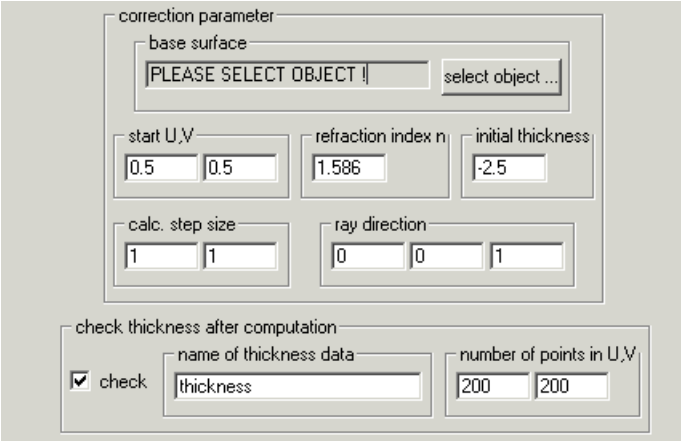
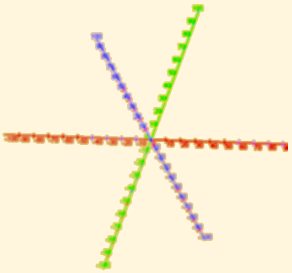


Figure 4.26: The Ray Deviation Correction dialog

4.10. Various Geometry Tools

4.10.1. Create Axis System

Creates a local axis system in Euclidean 3D coordinates. It can be used to define the position and orientation of geometric objects or a scaling in x-, y- or z-direction.



name	The identifier of the object.
axis arrow length	The length of the axis arrows in the Geometry View display.
origin x,y,z	The origin x,y,z of the coordinate system.
z-axis x,y,z	The z-axis of the system. The Vector (0,0,1) indicates that the z-axis is the optical axis. This axis points in the optical direction.
x-axis x,y,z	The x-axis of the system. The x-axis should be perpendicular to the z-axis, otherwise it will be corrected to be perpendicular.
sel...	Allows to select a Point3 to be used as the origin. According to z- and x-axis, only curves can be selected. They have to be orthogonal.
axis...	An existing axis system can be selected. The values for origin, z- and x-axis are taken over.
+/-	Changes from positive to negative values and vice versa.

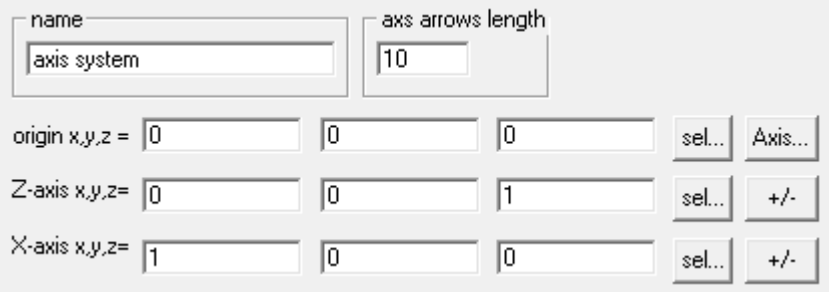
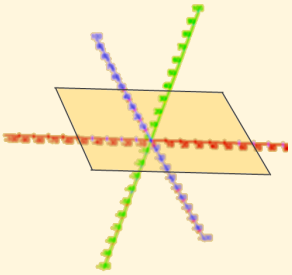


Figure 4.27: The Create Axis System dialog

4.10.2. Create Clipping Plane

Creates a virtual plane in order to clip objects in the geometry view.



normal (x,y,z)	The plane's normal vector for the orientation in the global axis system.
origin (x,y,z)	The origin of the plane within the global axis system.

4.10.3. Create Object Vector

Creates a new (empty) object vector in the current model. An object vector can be used as container to collect geometry data. (This is analogue to a folder or directory in a file explorer.) This command has no dialog.

4.10.4. Create Polylines from Rays

Captures rays from the GeoView and generates these rays as polylines, in other words, it creates a geometry (stored in an ObjectVector) These rays then can be used and exported to CAD systems. To delete polylines, use the TreeView or Geometry/Select/All Polylines and the delete button. The rays can be deleted as usual with the delete ray button. The length of the rays can also be controlled.

max. length of rays	Controls the length of the rays.
Create Polylines	Performs the creation of the polylines.

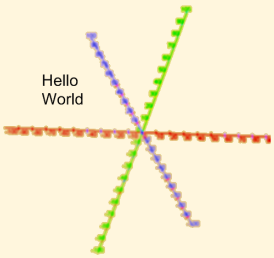
4.10.5. Create Cylinder from Rays

Captures rays from the GeoView and generates these rays as cylinders, in other words, it creates a geometry (stored in an ObjectVector). These rays then can be used and exported to CAD systems. To delete polylines, use the TreeView or Geometry/Select/All Polylines and the delete button. The rays can be deleted as usual with the delete ray button. The thickness and length of the cylinder rays also can be controlled.

thickness of rays	Controls the thickness of the rays.
max. length of rays	Controls the length of the rays.
create cylinder	Performs the creation of the cylinders.

4.10.6. Create Annotation Text

An annotation text can be created at a position defined by the origin in the geometry view.

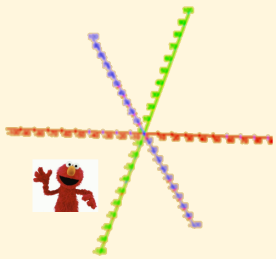


origin (x,y,z)	The position where the annotation text is created.
offset (x,y)	A pixel offset to fine tune the annotation's position.

annotation text	The annotation text, please note that country-specific letters and characters will not be shown in the GeoView (like e.g. 'ü', '§', '€', 'à', etc.)

4.10.7. Create Annotation Bitmap

An annotation bitmap can be created at a position defined by the origin in the geometry view.



origin (x,y,z)	The position where the annotation bitmap is created.
sel...	Opens a file selection window to select an existing image from the computer.
offset (x,y)	A pixel offset to fine tune the annotation's position.

Chapter 5. FreeForm Geometries Menu Selection

In this section, we treat the so called "Functional Geometry" that is available in the main menu. They are representations of selected design applications, making use of feature classes. The following commands are available under the menu entry FunGeo. We also cover special structures for (free form) surfaces, such as prisms or dot matrices which are used to create light pipes or backlighting devices.

5.1. The menu entries for FunGeo

- MF Reflector
- MF Lens

- Base Grid
- Axis Pattern
- Tube
- Lamp for Design Feature
- Sensor for Design Feature
- Switch Board for Design Feature

- PCS Reflector/TIR Lens
- PCS Reflector/TIR Lens with PS Curves
- PCS Reflector/Lens with PS Curves
- PCS Lens
- PCS Shield

- PS Reflector
- PS Lens
- PS Reflector IC
- PS Lens

- PS Reflector Curve
- PS Lens Curve

- Collimator LED Lens
- Prism Band for back light in light pipe
- Dot Mask for back light
- Retro Reflector

- Genetic Algorithm Optimizer
- Merit Function
- Parameter Space Explorer

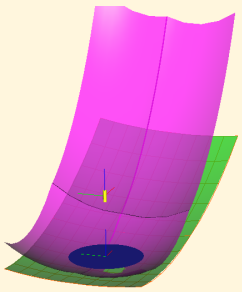
- Design Feature from old Config
- Model Note
- Model Documentation

A few main buttons and options are the same in all of the above FF geometries. These are listed in the following table:

Apply	Create the optical component using the current parameters.
Help	Opens up the appropriate chapter of a manual.
New Copy	Create a new entity of this FF-component, opening another dialog to specify the parameters of this second component separately. (Simple copies of an existing geometry can be made with copy & paste.)
Edit Axis	Edit the local axes system of the whole component (origin and orientation).

5.2. MF Reflector/Lens Surface

This FF geometry creates a reflector or a lens from a NURBS grid or a point grid, using the Macrofocal concept.



The material properties can be selected for a reflector or a lens (inner/outer optics). Reflectors require a coefficient of refraction to be specified (standard=1). Lenses are specified via refractive indices (inner, outer), the fresnel factor, and a scale factor. It is also possible to select a surface for a fix lens side.

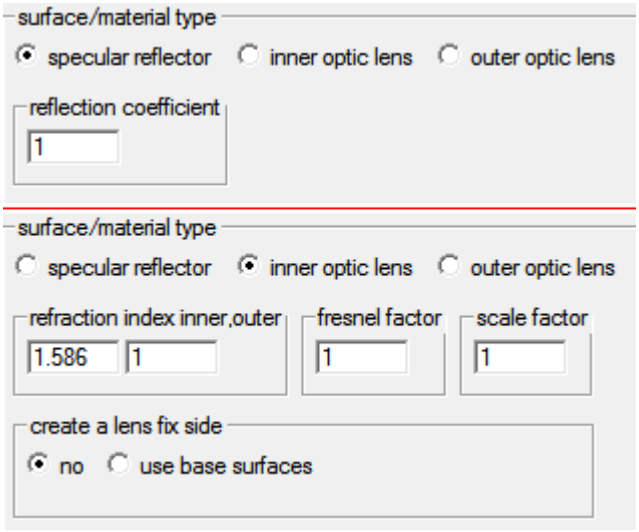


Figure 5.1: MF Reflector (Lens) Dialog: Reflective (Refractive) Material

In the grid layout settings, the base surface and its position are selected. Also, the number of facets is chosen here. You may mirror an incomplete grid (e.g. one quadrant only).

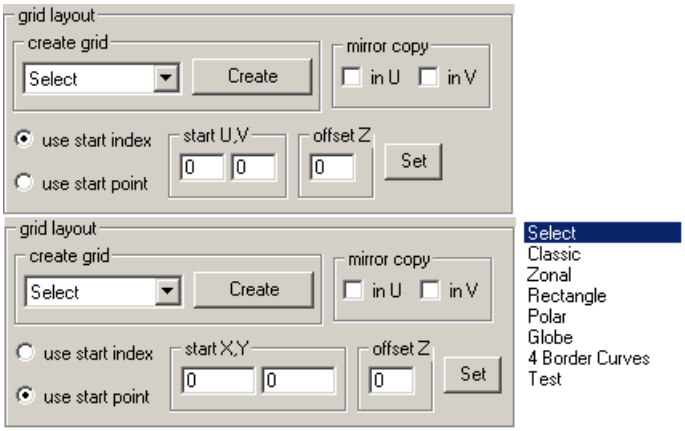


Figure 5.2: MF reflector / lens dialog: grid layout

The MF specific dialog buttons are listed below.

Figure 5.3: MF reflector / lens dialog: Dialog buttons

Gaps	Opens the dialog to define the gaps between the facets.
Spreads	Opens the dialog to define the spread angles from the component.
Source	Specify the source settings, including the selection of standard light sources, as well as the free definition of geometric sources or ray files.
F. Start	Opens the dialog to specify the facet start.
Switches	Opens a small dialog to activate/deactivate the facets individually.
Other Settings	Opens the dialog box for settings concerning the handling of patches, see below.

Please refer to the Macrofocal Application manual for the detailed properties and background information on the parameters of MF components.

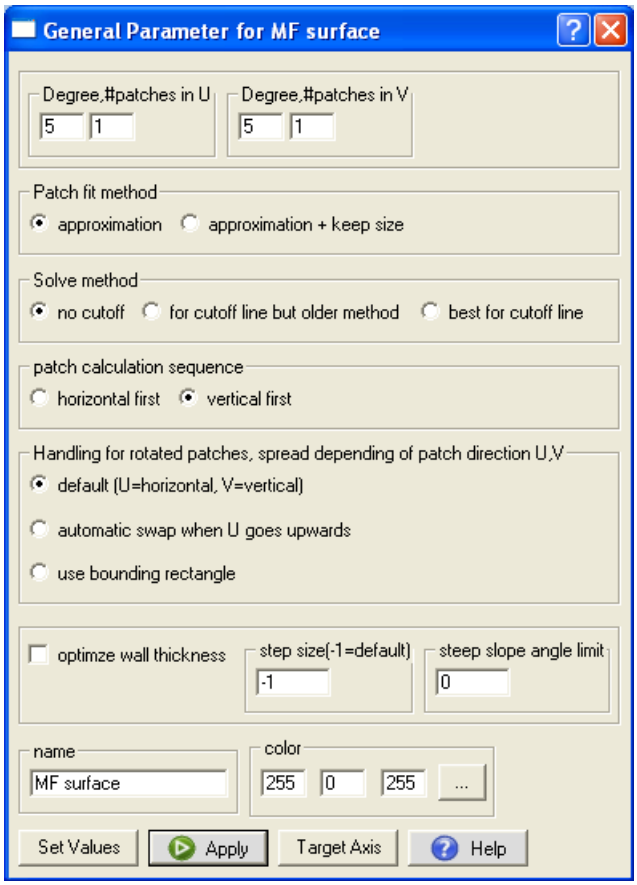


Figure 5.4: MF reflector / lens dialog: other parameters

With the button "other", you access the dialog for further parameters, such as listed in the table below.

Degree, #patches in U, V	Specify the NURBS degree and the number of patches in u- or v-direction.
Patch fit method	Select whether or not to keep size in approximation.
Solve method	Select the cutoff to solve for: no, cutoff line (older method), or best for cutoff line.
patch calculation sequence	Arrange the order of calculation: vertical first or horizontal first.
Handling of rotated patches, spread depending of patch direction U,V	Select how to deal with orientation: default (U=horizontal, V=vertical), automatic swap, or the use of a bounding rectangle.
optimize wall thickness	Select step size (standard: -1) and slope angle limit.
name	Select a name for the created shape.
color	Define the color (RGB).

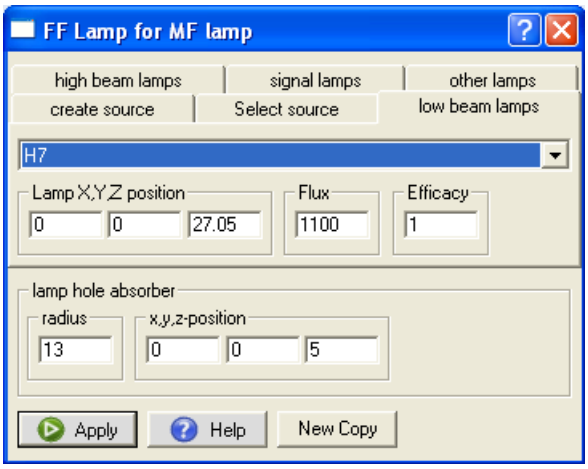


Figure 5.5: MF reflector / lens dialog: Set MF source

The Set MF source button triggers the Dialog for the free form lamp for MF the MF application. The six tabs allow the section from automotive lamps (high beam, low beam, signal, other). Also, a source can be selected or created in this dialog.

Lamp X,Y,Z position	Place the light-source in the cartesian system.
Flux	Specify the luminous flux of the source. This will default the values from a chosen lamp model, but can be altered manually.
Efficacy	Specify the efficacy (0..1). This can be used to account for specific as well as overall losses.
lamp hole absorber	Radius and location of an absorber disc representing the lamp hole and socket.

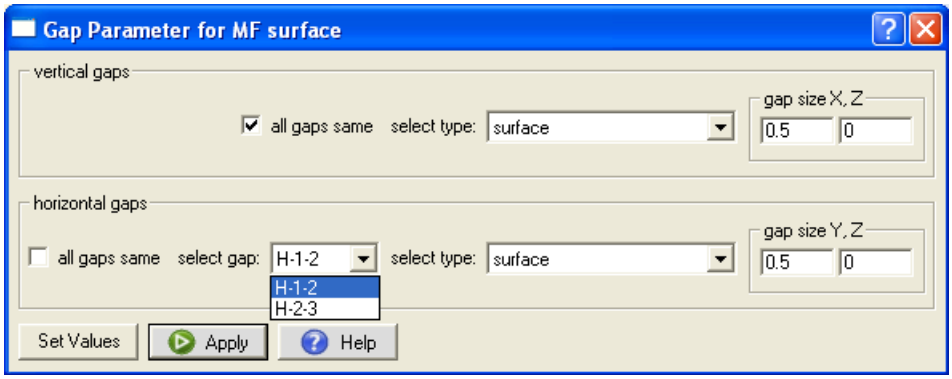


Figure 5.6: MF reflector / lens dialog: Set Gaps

To specify the gaps between facets and also their connection types (please consult the Macrofocal Application Manual for details), the "Set Gaps" button opens a new dialog window. You specify the horizontal and vertical gaps separately.

all gaps same	Check this box if you have more than one gap, but you want to use the same gap type for all gaps (in that direction).

select gap	This selection appears only, if you have more than two facets in a direction (else, there will be only V-1-2 or H-1-2). To select the gap between the two vertical facets 2 and 3, chose "V-2-3".
select type	The type of facet gaps connection is chosen here. Please take a look into the Macrofocal Application Manual for reference.
gap size X,Z / Y,Z	Specify the 2D gap size.

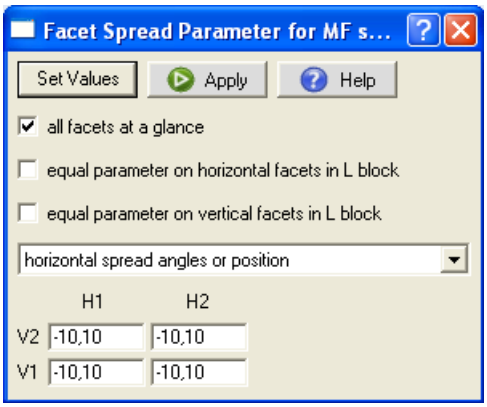


Figure 5.7: MF reflector / lens dialog: Set Spread

To specify the spread angles from the facets, the appropriate dialog gives access to the values for each facet.

all facets at a glance	Check this box to see the reduced representation of the spread angle dialog, showing only the spread values for quick access.
equal parameters on horizontal/vertical facets in ( ) block	If you do not need individual spread angles, check this box to apply the same spread to all facets of one block
facet parameters	Horizontal and vertical spread angles are defined in degree of angle [deg].
target type	Selects the axis system (e.g. Y-pole)
Macrofocal edge ray use	Chose max, min, or center as the location of the cutoff line in the specific direction (e.g. top, bottom, or center).
cutoff angle	Define a fix slope of the cutoff line in the beam pattern.

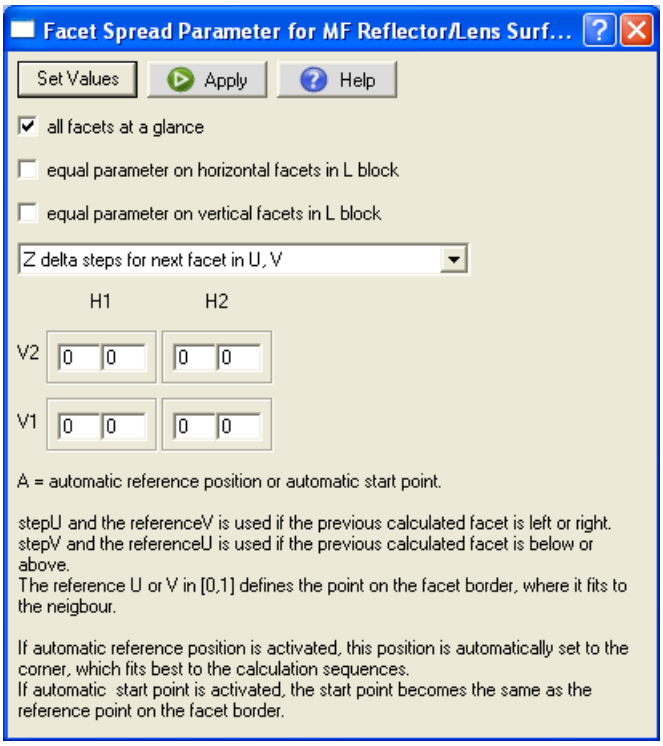

Facet Spread Parameter for MF Reflector/Lens Surf...
Set Values
Apply
Help
all facets at a glance
equal parameter on horizontal facets in L block
equal parameter on vertical facets in L block
Z delta steps for next facet in U, V
H1
H2
V2
V1
0
A = automatic reference position or automatic start point.
stepU and the referenceV is used if the previous calculated facet is left or right.
stepV and the referenceU is used if the previous calculated facet is below or above.
The reference U or V in [0,1] defines the point on the facet border, where it fits to the neigbour.
If automatic reference position is activated, this position is automatically set to the corner, which fits best to the calculation sequences.
If automatic start point is activated, the start point becomes the same as the reference point on the facet border.

Figure 5.8: MF reflector / lens dialog: Set facet start

The Set F. start button gives access to a dialog window to tune the calculation of the facets (star, reference, Z-steps)

all facets at a glance	Check this box to see the reduced representation of the dialog, for quick access. This alters the complete appearance of the dialog.
equal parameters on horizontal/vertical facets in ( ) block	If you do not need individual spread angles, check this box to apply the same spread to all facets of one block.
Z delta steps for next facet in U,V	Define the steps between the facets (in Z direction), perpendicular to the reflector. Steps can be specified separately for U direction (horizontal facets) and V direction (vertical facets).
reference position in [,] along facet edge U,V	Define the border position with U,V coordinates to control the connection of a facet to its neighbor. This function is influenced by the chosen patch calculation sequence. In case of individual facet treatment, you can switch to automatic reference with a check-box.
calculation start point in [,] U,V	Define the start point for the calculation of a facet. These values can be altered manually to assist when default positions (o,o) lead to facet degeneration or other strange calculation results. In case of individual facet treatment, you can switch to automatic start point with a check-box, this uses the reference point (see above) as start point.

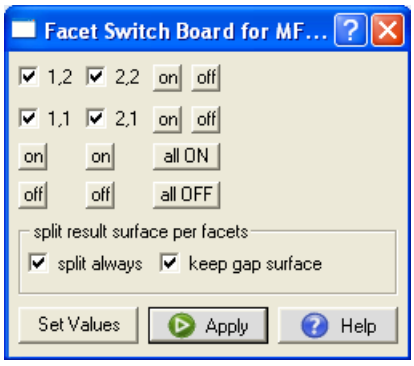


Figure 5.9: MF reflector / lens dialog: Switches

The active facets can be chosen by switching each facet on or off.

on, off	Turn a facet on or off by (un-)checking its box. The facets are labeled in order of position (U,V).
split always	Treat facets individually, e.g. for sensors.
keep gap surface	Take surfaces between facets into account in simulations. Uncheck to ignore gap surfaces.

5.3. Switch Board

When a model grows larger and contains more than one or two light functions, it becomes harder to keep the overview which emitter and sensor belong to which design feature. So for every simulation of a different light function one has to browse the DesignFeature folder in the TreeView and manually set/toggle all the traceable flags.

The main intention of the switch board is to group certain design features into a set. This set now can be controlled with only a few mouse-clicks within the dialog (controlling means to set the flags T and/or V).

The dialog right below shows an exemplary setting with a total of three sets. There are one PCS and one MF set, both containing the main feature, an emitter and a sensor. The third set is just to show all features at the same time.

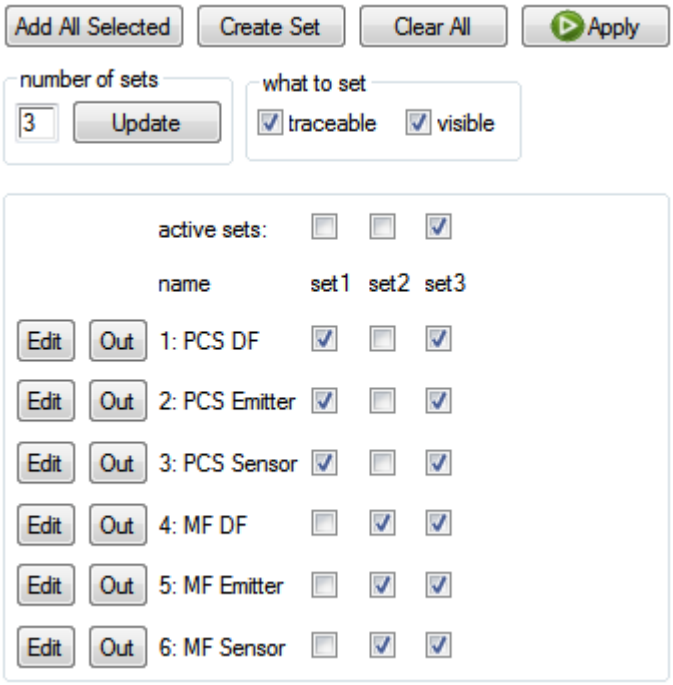
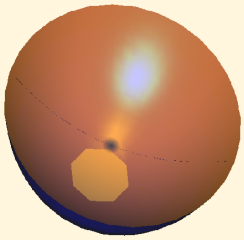


Figure 5.10: Switch Board for Design Feature

button: Add all selected	Mark an entry in the DesignFeature folder in the TreeView and press this button to add this feature to the switch board.
button: Create set	After all wanted features have been added into list and all settings have been done, press this button to create the sets, i.e. to determine which features shall be active at the same time (grouped).
button: Clear all	If pressed, all features listed in the switch board are set to invisible in the GeoView.
button: Apply	If pressed, the current settings are applied onto the model.
number of sets	Determines the number of columns in the table, i.e. the number of sets the features can be grouped into.
button: Update	If pressed, the number of columns are updated and the names of the features are adopted from the model.
what to set	Using these two flags one can determine which shall be done with an inactive feature (i.e. a feature which is not in an active set).
button: Edit	If pressed the feature dialog will open up.
button: Out	If pressed the feature is removed from the list (but not from the model).
flag: Active Sets	These check-boxes are used to determine which set shall be set as active. Please note that only one set per time can be set as active.

5.4. PCS Reflector/Lens Surface

This FF geometry creates an Poly Curve System from specified base curves. Per default these are four curves, named *West, North, East* and *South*. The PCS can be used to create either a reflector or a lens. It allows you to define the number of facets and their individual spread.



Like all PCS systems, the main curves are defined by their origin, angles (orientation within the local xy-plane), start points and the length of each section (profile) of each curve. The shape of the component in between these curves is defined by the weights (NURBS).

For a more detailed description please also refer to the separate manual ProceduralSurfaceApplications.pdf (accessible via either the help menu or in the doc folder of your installation of LucidShape).

The main dialog contains all the buttons for the further setting. Pressing any of the buttons (except *Apply* or *Help*) will open a further dialog for the sub-settings.

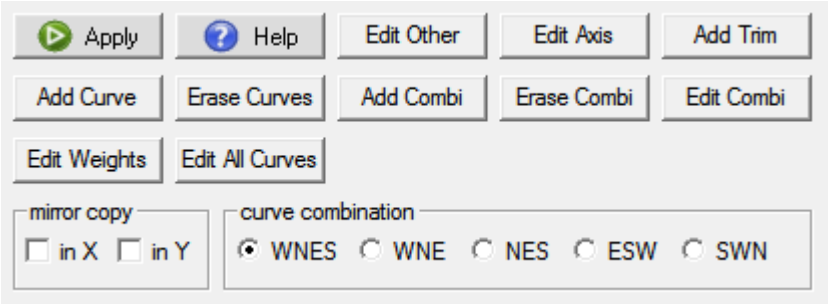


Figure 5.11: PCS Main Dialog

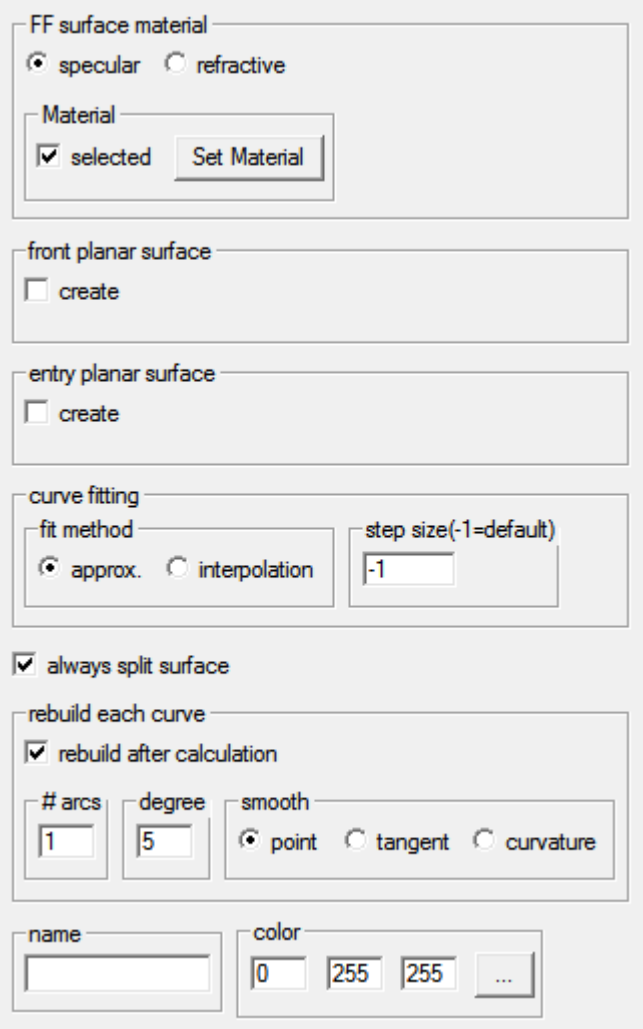


Figure 5.12: Edit Other Dialog

radio button: FF surface material	Defines whether the surface is either specular or refractive.
button: set material	Pressing the button allows to select a material from the library.
reflection coefficient	Defines the amount of light being reflected, from 0 (nothing) up to 1 (perfect reflector).
refractive index, fresnel factor, extra scale	Specifies these three parameters for a refractor only.
flag: make outer medium total absorbing	Selects whether the outer surface will be a perfect absorber or not (e.g. for absorbing scattered light).
flag: front/entry planar surface	En-/disables the parameters to create a front or entry surface.
center position	Sets the potion of the front or entry surface of the reflector.
weight	Specifies somehow the size this surface (whether it is curved or flat).
flag: always split surface	The reflector surface is split into its facets. When performing a simulation using the *Light Mapping* method, it is possible to observe the simulation result for each single facet. This way their respective contribution to the whole LID can be examined. Choose

	"Navigate in 3D/4D data set..." in the context menu of the light data window to navigate through the different facets.
flag: rebuild after calculation	Makes a special update of the current settings to the model in focus.
# of arcs	Defines the number of arcs for one facet in each direction.
degree	Defines the polynomial degree of the two base curves.
radio button: smooth	Selects the type of smoothing for the curves.
name, color	The name and color will be set (e.g. for differentiation).

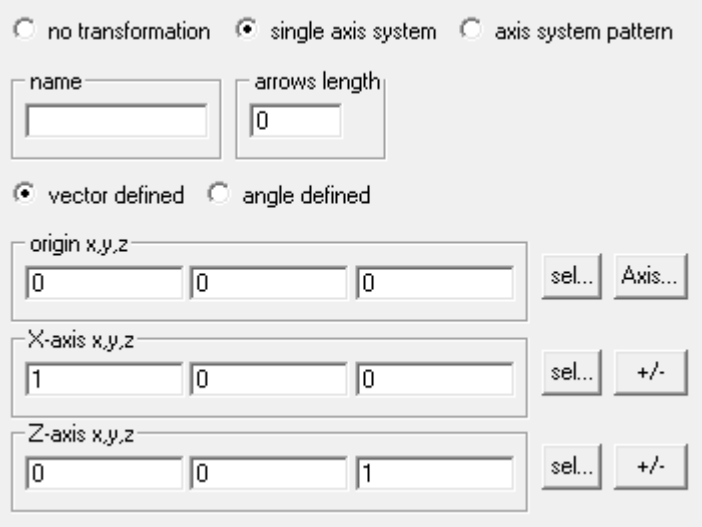


Figure 5.13: Edit Axis Dialog

radio button: no transformation, single axis system, axis system pattern	Selects the type of the transformation: No transformation at all, a transformation about a single axis or onto an axis system pattern.
name and arrow length	Sets the name and the arrow length.
radio button: vector/angle defined	Defines if the parameters for the transformation are vector or angle defined.
origin x,y,z	Defines the position of the plane.
x-axis x,y,z	Defines the orientation of the x-axis in the setting.
z-axis x,y,z	Defines the orientation of the x-axis in the setting.
rotation angles about Z, Y', X''	Defines the values of the angles for the rotation of the plane. (Please note: the order in which the rotations take place is important, with rotation about the z-axis the x-axis and y-axis are transformed into x'-axis and y'-axis, with rotation about y' : x'→x".)
select feature	Selects an (already existing) axis pattern from the geometry.

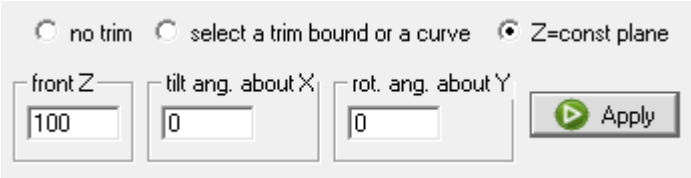


Figure 5.14: Add Trim Dialog

radio button: no trim, select a trim bound or a curve, z=const plane	Selects the type of the trim: no trim at all, about a trim bound or a curve, or about a plane with a fixed z-coordinate.
select trim bound or a curve	Selects the bound/curve at which the trim shall take place.
front z	Defines the z-coordinate of the trim plane.
tilt angle about x,y	Sets the tilt angle(s) about the x-/y-axis for the trim plane.

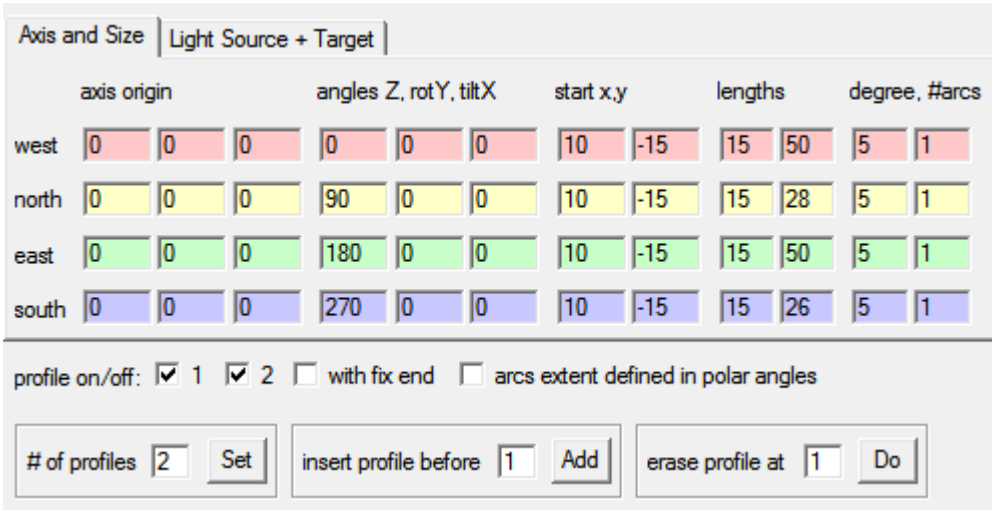

Axis and Size
Light Source + Target
axis origin
angles Z, rotY, tiltX
start x,y
lengths
degree, #arcs
west 0 0 0 0 0 0 10 -15 15 50 5 1
north 0 0 0 90 0 0 10 -15 15 28 5 1
east 0 0 0 180 0 0 10 -15 15 50 5 1
south 0 0 0 270 0 0 10 -15 15 26 5 1
profile on/off: 1 2 with fix end arcs extent defined in polar angles
# of profiles 2 Set
insert profile before 1 Add
erase profile at 1 Do

Figure 5.15: PCS Dialog: Axis and Size Part

axis origin	The (x,y,z)-tuple to determine the axis origin of the single curves, they can be set independently to each other.
angle z	This is the angle respective to the x-axis, the default setting is a distance of 90° between two neighbouring curves.
rotY	Determines the angle around the y-axis, e.g. a positive angle shifts the curve towards the z-axis.
tiltX	Determines the angle around the x-axis.
start x,y	Defines the starting coordinates for the curve creation.
lengths	Defines the length of the arcs.
degree	Sets the polynomial degree of the individual profile arcs.
# arcs	Defines the number of arcs to be fitted. The curve may be rebuilt after calculation with the entered degree and arcs in order to obtain a more smooth reflector, but keep in mind that this may cause modifications in the light function.

flag: profile on/off	Switches one or both profiles on/off.
flag: with fix end	En-/disables a fix end of the reflector's surface and one can change this fix end via the two values in the table.
flag: arcs extend defined in polar angles	If disabled, the extent is given in cartesian coordinates. If enabled, the values determining the extent are given in degrees.
# of profiles	Changes the number of available profiles.
insert profile before	Specifies the position where the new profile is inserted.
erase profile at	Specifies the profile to be removed.

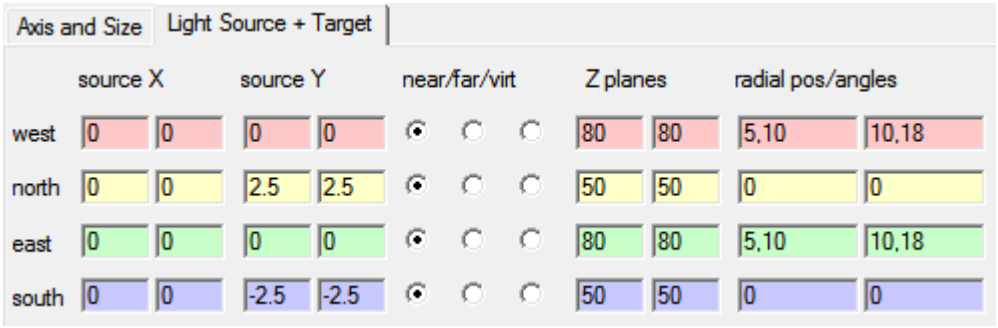
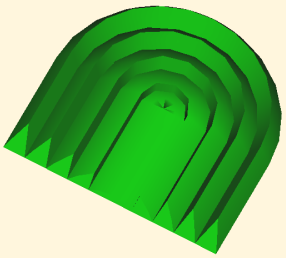

Axis and Size
Light Source + Target
source X
source Y
near/far/virt
Z planes
radial pos/angles
west 0 0 0 0 80 80 5,10 10,18
north 0 0 2.5 2.5 50 50 0 0
east 0 0 0 0 80 80 5,10 10,18
south 0 0 -2.5 -2.5 50 50 0 0

Figure 5.16: PCS Dialog: Light Source and Target Part

source x,y	Defines the light starting position for the corresponding base curve.
radio button: near/far/virtual	Switches between near field (the entered values are interpreted as a position on the z-plane), far field (the values stand for outgoing angles) and virtual (the values stand for a position on a virtual z-plane).
z planes	The z-plane for the creation of the profile arcs is the focal plane of the individual base curve profile. The values entered are the distances from the origin to the target planes. The number of entries in the dialog box corresponds to the number of profiles of every base curve. Therefore, every profile can have a different target z-plane position. This is the position of a virtual plane where the light will be focused by the individual parts of the reflector.
radial positions/angles	The target radial positions or angles for the creation of the profile arcs are the focal points on the target plane. The number of input parameters (radial positions or angles) corresponds to the number of profiles. These entries correspond to x,y,z-coordinates on the plane. The radial position is the position on the target plane where the reflected light will be focused.

5.5. PCS Lens

This FF geometry creates a poly curve system lens. Besides aspherical lenses, also TIR or Fresnel optics can be generated.



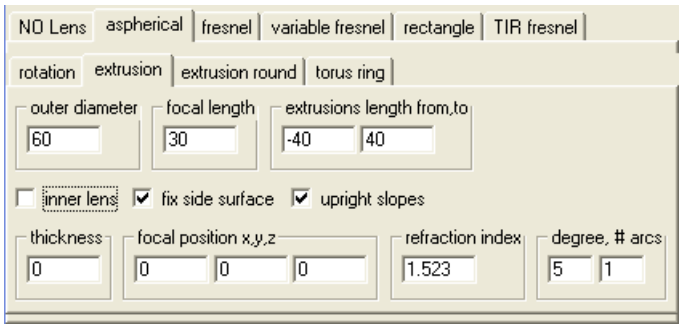


Figure 5.17: FF Geometry - PCS Lens Dialog: round aspheric lens

In contrast to the "lens surface", the PCS lens is a complete component, consisting of both the inner and outer surface. Tabs in the dialog allow to chose between the following types:

- aspherical
- Fresnel
- variable Fresnel
- rectangle
- TIR Fresnel

Common parameters of most lenses are:

focal length	Specify the focal length of the lens
thickness	An overall base thickness which is added to structural features of the lens (like curvature or riffles).
inner lens	Check to place all structure on the inside of the lens (pointing towards the source), outer side is planar. Uncheck to invert.
fix side surface	Uncheck to deactivate the other surface (outer/inner) of the lens.
upright slopes	Selects an algorithm which creates upright slopes instead of slanted ones. Difference is stronger at larger thicknesses.
focal position	The location of the focal point (x,y,z) determines the position of the lens, keeping it at focal length.
refraction index	The index of refraction of the lens material, is applied to all interfaces (surfaces of the lens).
degree	Polynomial degree of a lens curve (arc).

# arc	Number of curve segments (arcs).

Extrusion type parameters

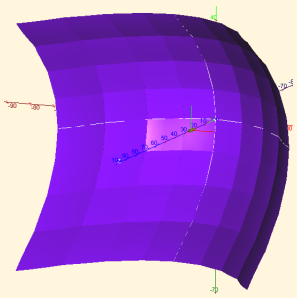
extrusions length from, to	Specify the boundaries of the extrusion start, stop i the local axis system.

Torus ring type parameters

start, sweep angle	Starting angle and sweeping range of the lenses arc.
focus radius	The focal radius is added to the radii of the lens.

5.6. Procedural Surface (PS) Reflector

This group of FF geometries creates a PS Reflector with different base forms. It allows the user to define different focal positions for the horizontal (H) and vertical (V) curvature of the reflector.



The concept of Procedural Surface Application (PSA) is used for both reflectors and refractors in LucidShape. The respective dialogs have several identical options and parameters.

For a much more detailed description of the options, parameters and a short tutorial, please see also the separate manual *ProceduralSurfaceApplications.pdf*, available in the same folder as the other pdf-files (as per default ..\LucidShape\doc\) or via the help-button in the respective menu entries.

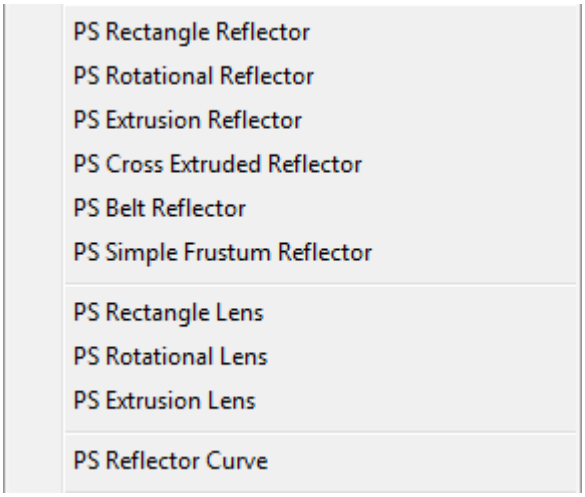


Figure 5.18: FF Geometries PS Reflectors (in the FunGeo menu)

PS Reflectors are created from facets that can be oriented and shaped individually. There are seven basic shapes of this type in LucidShape, which only differ in their specific geometries.

Just like in the creation of PCS geometries, also in PS components the spread angles of individual parts of your optical component can be defined. Here, each profile (like a facet) can be assigned the desired light spread angles.

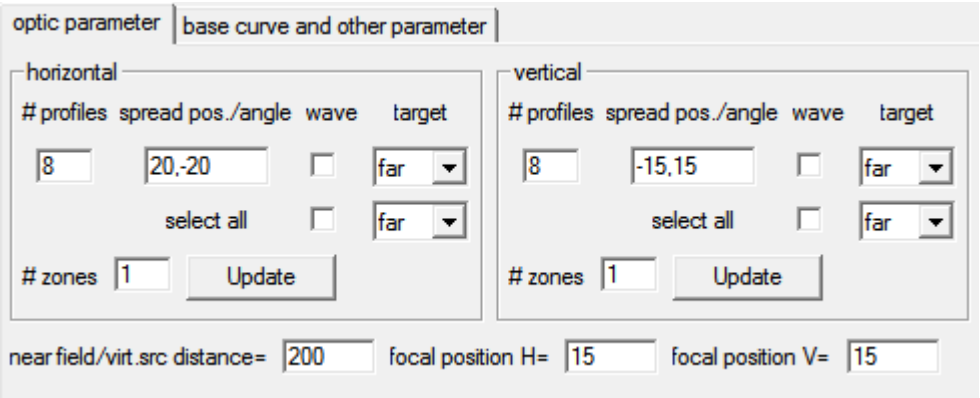


Figure 5.19: PS Reflector Dialog: Part with Optic Parameters

# profiles	Defines the number of profiles. They can be set both for horizontal and vertical direction for each zone.
spread pos./angle	In the case of using near field and virtual, these entries define the position on the plane positioned at near field/virtual distance. Otherwise they describe the spread angle for each profile (in both cases for each zone).
flag: wave	If checked, the profiles are created alternately up and down, otherwise they all are formed convex (towards the inner of the reflector). This flag can be set independently for the horizontal and vertical profile.
target	Allows to select among near field, far field and virtual. This choice determines the meaning of the values entered for *spread position/angle*.
(flag) select all	Both the *wave* flag and the target can be set for all profiles at once.
# zones	The number of zones to be created, this does not change the size of the reflector.
button: update	After every change of the number of zones, this button has to be pressed to update the dialog.
near field/virtual source distance	The distance of the near field plane or the virtual source, respectively. When using far field, this parameter is ignored.
focal position H,V	Defines the focal positions for both horizontal and vertical.

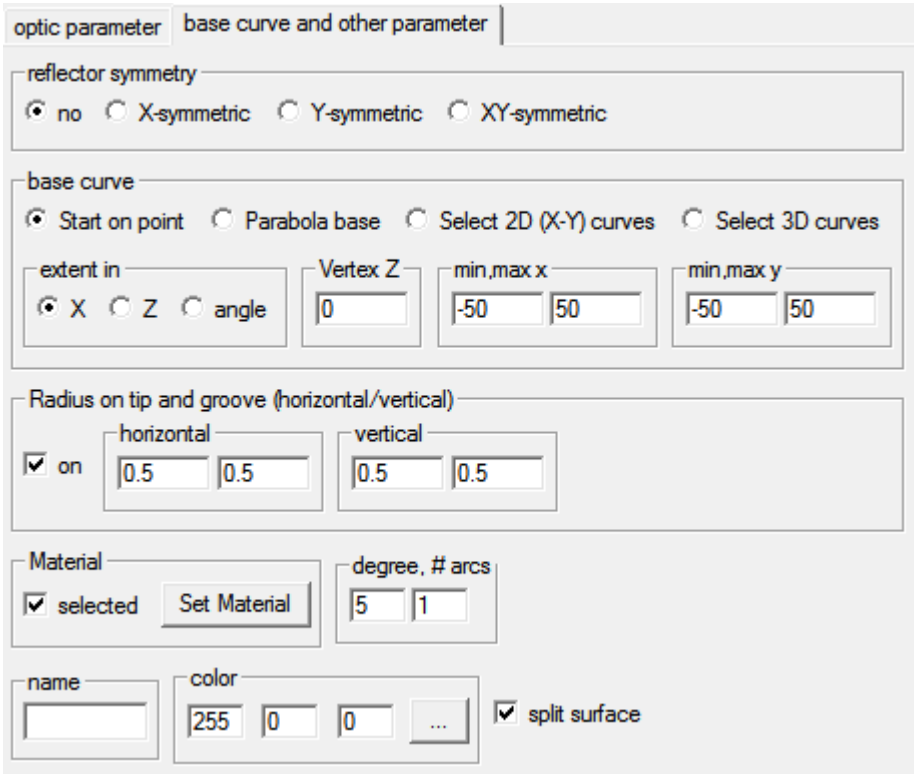


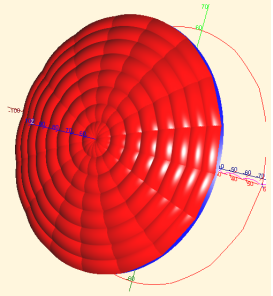
Figure 5.20: PS Reflector Dialog: Part with Base Curve Parameters

radio button: reflector symmetry	Determines types of symmetry the base values shall have: Whether no symmetry at all, symmetrically with regard to the x-axis, the y-axis or with regard to both of them.
base curve: start on point	The reflector calculation starts on one single point.
radio button: extent in	Determines the calculation 'direction' used for the reflector's calculation. This is either in x-, in z-direction or via angle. The direction can be changed to avoid some mathematical problems, like the handling of poles at the origin of the reflector when using the usual extent in x.
vertex z	The reflector's vertex position on the z-axis.
min/max x, min/max y	Determines the size of the rectangular reflector in x,y-direction.
flag: radius on tip and groove	Similar to the feature available for prism-band, an optional radius can be added onto the tip and/or groove of the PSCurves. This will round the formerly (more or less) sharp edge of the resulting shape. The values can be set independently to each other.
button: set material	Pressing the button allows to select a material from the library.
reflectance	Defines the reflectance, the value has to be in the interval [0,1].
degree	Defines the polynomial degree of the two base curves.
# of arcs	Defines the number of arcs for one facet in each direction.
name	The name of the created reflector surface.
color	The display color of the reflector.

flag: split surface	The reflector surface is split into its facets. When performing a simulation using the *Light Mapping* method, it is possible to observe the simulation result for each single facet. This way their respective contribution to the whole LID can be examined. Choose "Navigate in 3D/4D data set..." in the context menu of the light data window to navigate through the different facets.

5.7. Procedural Surface (PS) Refractor

This group of FF geometries creates a PS Lens with different base forms. It allows the user to define different focal positions for the horizontal (H) and vertical (V) curvature of the refractor.



The concept of the PS Lenses is the same as for the PS Reflectors, the available options and parameters are nearly the same as for their reflector counterparts, the main difference are additional options and parameters for refraction (like refraction index, thickness, inner/outer optic, etc.) in the respective dialogs.

For a more detailed description of the options, parameters and a short tutorial, please see also the manual *ProceduralSurfaceApplications.pdf*, available in the same folder as the other pdf-files (as per default ..\LucidShape\doc\) or via the help-button in the respective menu entries.

There are three base shapes available for PS Refractors in LucidShape: Rectangle Lens, Rotational Lens and Extrusion Lens.

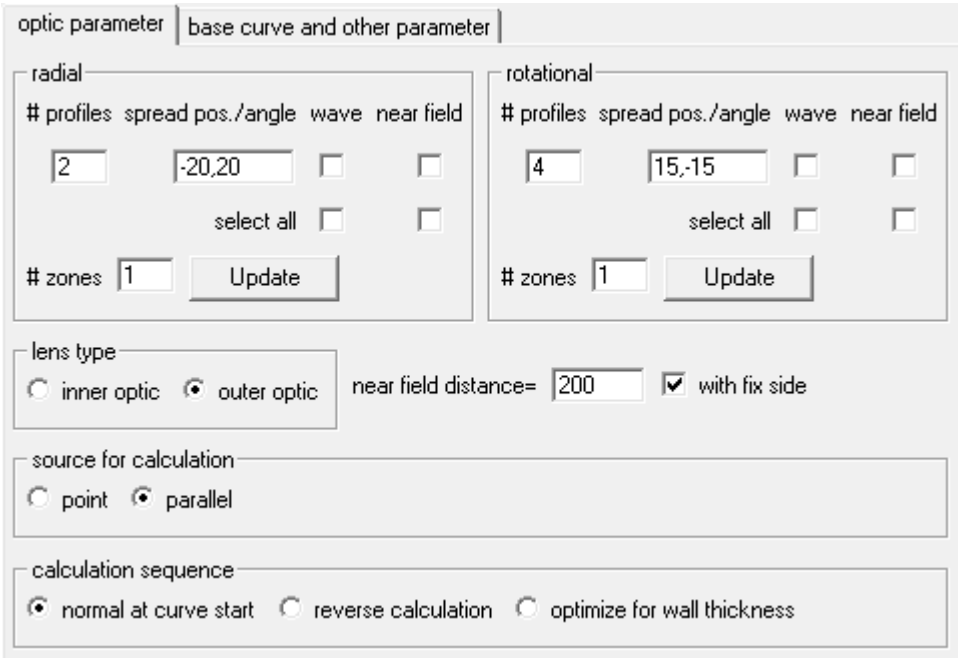


Figure 5.21: PS Lens: Dialog part with optic parameters

radio button: lens type	Determines whether this lens shall be an inner or an outer optic.
refraction index	Determines the refraction index of this material.
fresnel	Determines whether the refractor is ideal (without any reflection) or with the Fresnel law taken into account.
scale	The scale allows to take losses due to absorption into account without having a physical model of the material. 1 is equal to perfect transmissibility, 0.5 reduces the intensity of each ray to the half of its actual value (the total number of rays stays the same).
thickness	Determines the thickness of the material.

The other options and parameters are (nearly) identical to those of the PC Reflector.

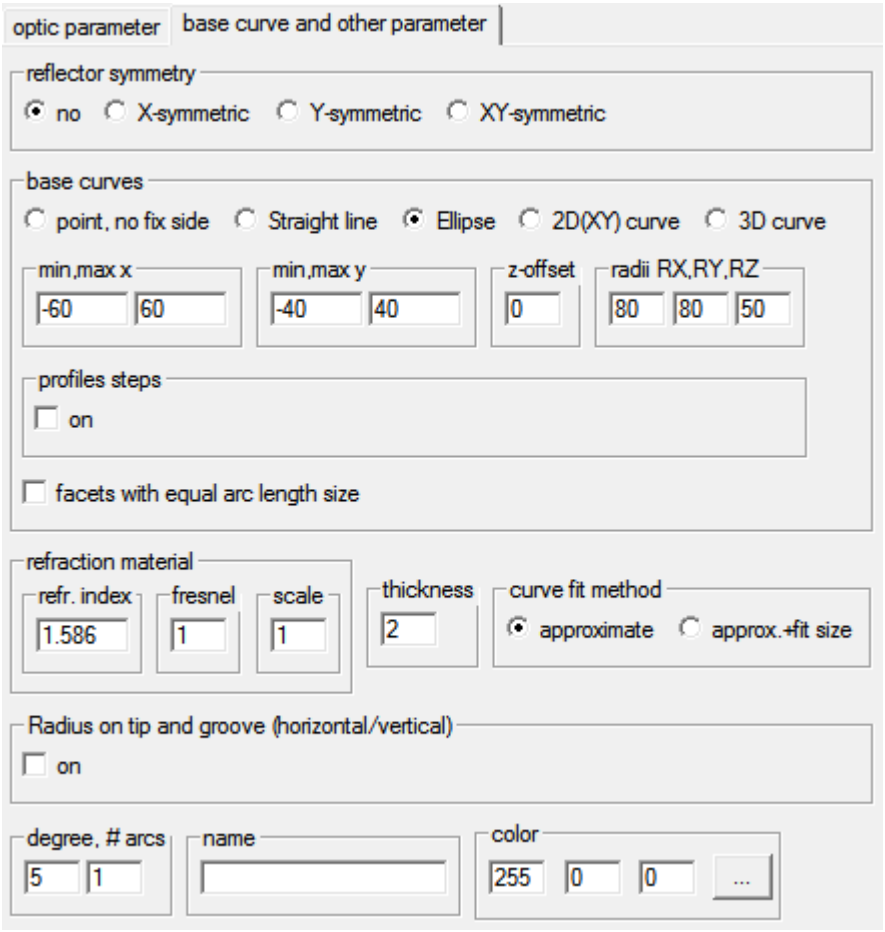
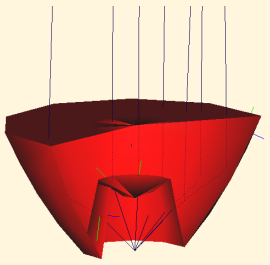


Figure 5.22: PS Lens: Dialog part with base curve parameters

5.8. Collimator LED Lens

This FF geometry creates an LED collimator (also concentrator) optical part to be mounted directly on top of an LED primary optic. You can choose between a rotational symmetry (round symmetric, one curve), elliptical (two different main curves) or specify the curves (N.W.S.E.).



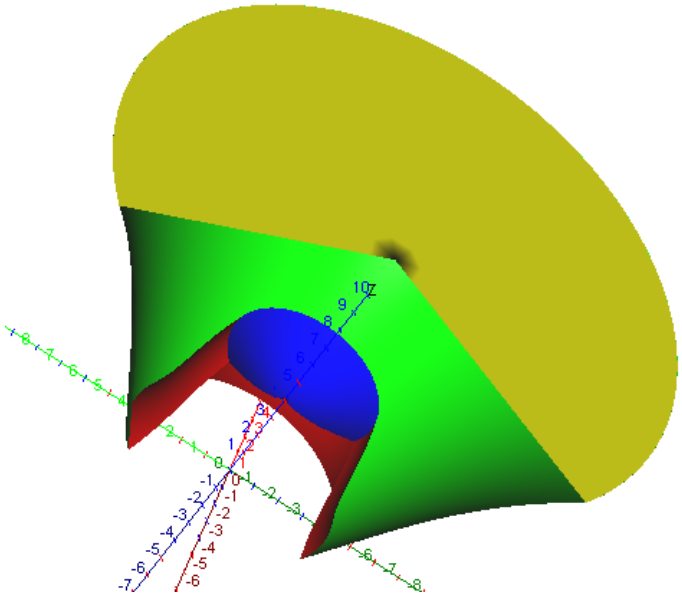


Figure 5.23: The parts of a LED Collimator

The whole collimator contains several parts, mainly there are the center (inner part) and the reflector (outer part), both of them can have a front shape independently. In the picture just above there are the front center (blue), the side center (red), the reflector (green) and the front shape (yellow).

In case of a round symmetric collimator, two optical interfaces must be defined: The front entrance and the side entrance. Light coupled in via the front entrance experiences refraction and propagates through the medium of the collimator. Light passing though the side entrance is thrown at the reflector where internal reflection occurs at the refractive index interface with the surrounding air.

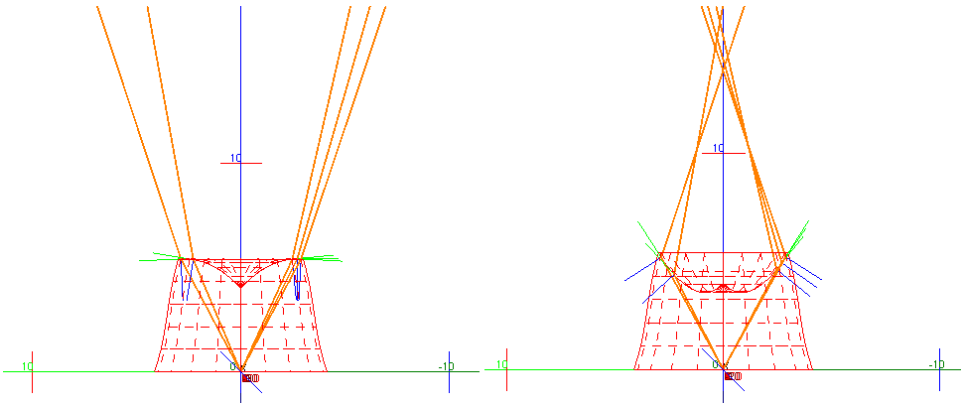


Figure 5.24: The difference between divergent (left) and convergent light (right)

In the two figures Figure 5.23, “The parts of a LED Collimator” and Figure 5.25, “Parallel (collimated) Light” one can see how the spread parameters are working and the difference between divergent and convergent light. In the left case the parameter values '20,-20' was used, so the light is spread from 20° (left side of the left picture) to -20° (right side of the left picture), this is called divergent. On the right side the parameter values are '-20,20', the resulting light behaviour is called convergent because the rays are crossing each other after exiting the collimator.

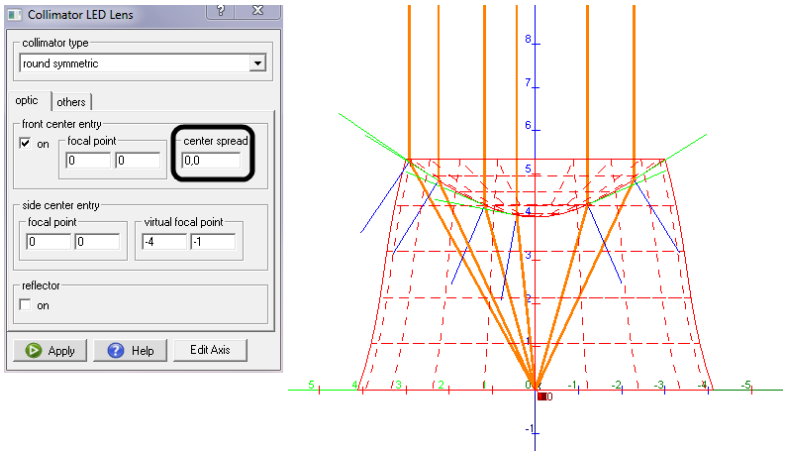


Figure 5.25: Parallel (collimated) Light

With the values for the center spread set to zero, the front center collimates the outgoing light to a parallel ray bundle (they are spread homogeneously from 0° to 0°).

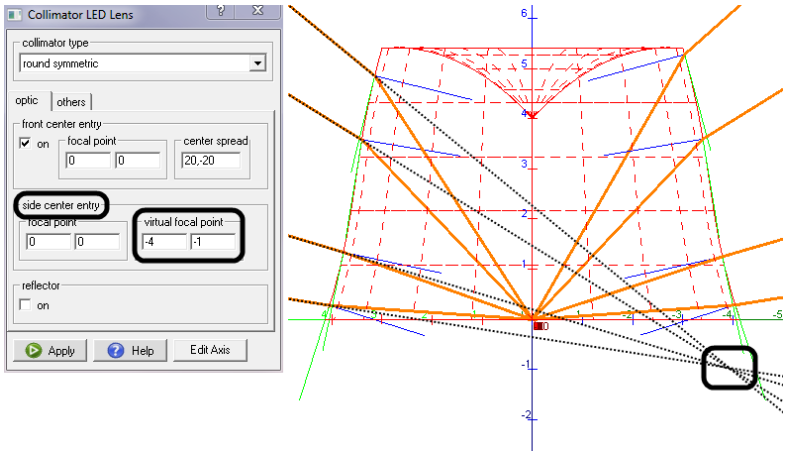


Figure 5.26: The virtual Focal Point

All rays start at the center of the collimator at (0,0,0) (solid orange lines) and are deflected by the side center. If these deflected rays would be traced back, from outside the object without the deflection (dotted black lines), they are heading to one point, the virtual focal point (marked in black). This point is called virtual because only from the outside the collimator this point is seen as the emitting point.

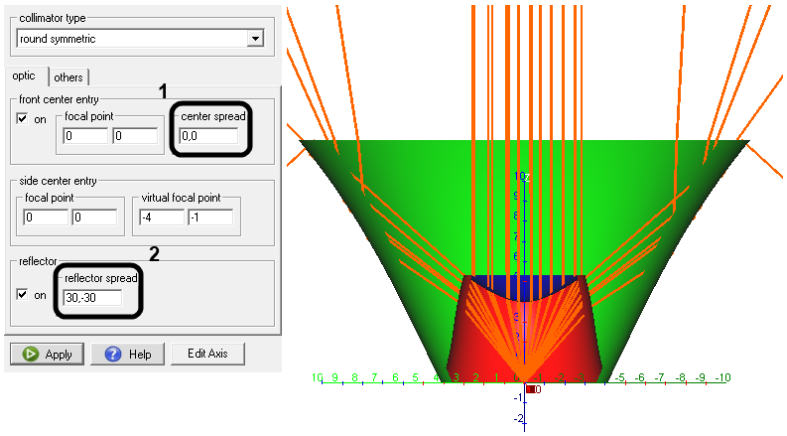


Figure 5.27: The whole Collimator with Center and Reflector

Here one can see both the Center and the Reflector and their respective light deflection. The values for the spread angles of the center are set to '0,0' (1), thereby the rays outgoing from the front center form parallel light. The rays outgoing the side center are spread by the reflector in a range of 30° to -30° due to the parameters entered for the reflector (2). The opposite way does work as well, just change the parameters for the center and the reflector. As a result the outer part of the front shape (yellow) emits parallel light, while the inner rays are spread from 30° to -30°.

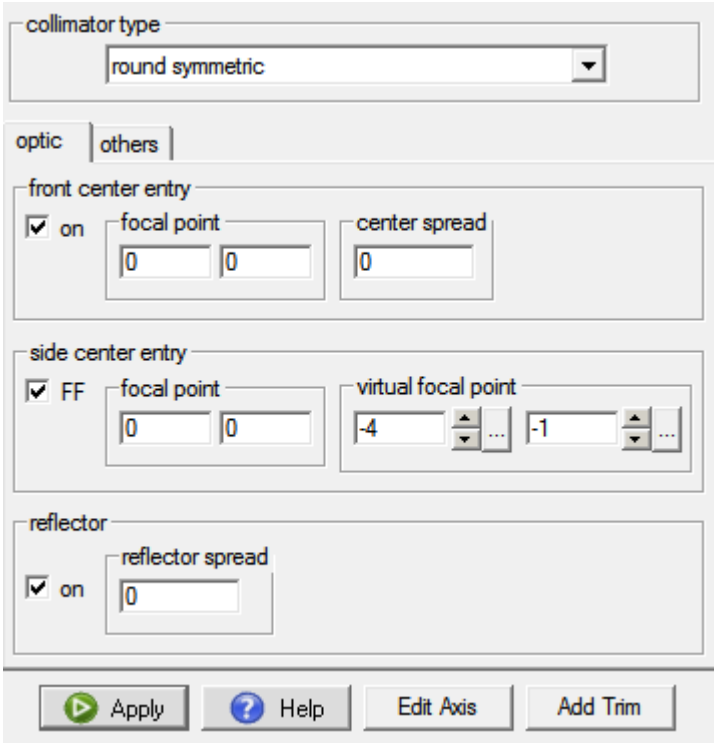


Figure 5.28: The LED Collimator Dialog, tab for optic parameters

flag: front center entry	Determines whether a front shape shall be added to the center or not.
focal point	The focal point of the curved front center entry. This should fit the emitter's center (alignment).

center spread	The light is spread in a range from first_value to second_value, in case of a zero spread, a collimator will be created which emits parallel light.
focal point (side center entry)	In most cases these values are identical with the focal point for the front center entry.
virtual focal point (side center entry)	The curved side entry illuminates the internal reflector, a second (virtual) focal point is created which is effectively seen by the reflector.
flag: on	Determines whether the collimator shall contain a reflector (e.g. this can be used to do the settings for the center first while having the reflector faded out).
reflector spread	The values entered here determine the spread angle(s) for the reflector part.

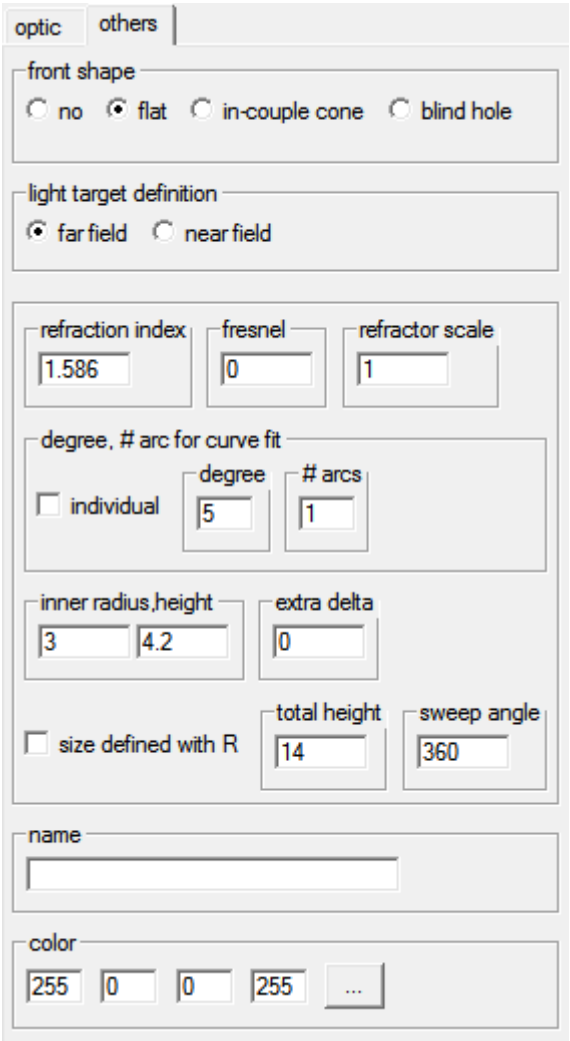


Figure 5.29: The LED Collimator Dialog, tab for other parameters

radio button: front shape	You can select the front shape (exit aperture) of the collimator: • nothing (to use other optics, such as a pillowed lens) • flat (standard) • in-couple cone (e.g. for fiber-coupling, with specified radii) • blind hole (mainly to save material, a frustum of a cone is cut out of the front shape; the angle always is oriented towards the centre line of the collimator)
radio button: light target definition	Toggles between far field and near field (with a given distance to the target plane).
refraction index	Specify this value according to your material (e.g. PS, PMMA).
fresnel	Determines whether the refraction shall be according to Fresnel's law or not.
refractor scale	Determines the refractor scale.
flag: individual	If checked the degrees of the center, side and reflector part can be chosen independently.
degree	Determines the NURBS' degree of the curve.
# arcs	Determines the number of arcs the respective surface is made of.
sweep angle	Beginning from the start, the shape is swept by the sweep angle value (e.g. a value of 120° creates a third of a complete circle).
inner radius, height	Sets the dimension of the collimator to fit the dimensions of the LED.
extra delta	Keeping the default value of zero, the collimator is created with a perfectly sharp edge at the bottom. Increasing this value results in a plane plateau between the Center (red) and the Reflector (green).
flag: size defined with R	This flag can be used to specify the extent of the resulting collimator either by the height (default setting, flag unchecked) or by the radius (checked flag).
total height/radius	This is the overall height/radius of the collimator, e.g. to take a limited maximal size into account.
flag: with curve (extrusion)	If checked it is able to choose an existing curve as the extrusion curve, otherwise the two values '*extrusion from, to*' specify the starting position and the length of the front center.

The base collimator type is *round symmetric*, this is what we have been used until now. The other available types are *elliptical, Poly Curve System types (PCS), extrusion symmetric, extrusion PCS, rectangle symmetric* and *rectangle PCS*.

In case of an elliptical collimator, there are two main curves to be defined. Thus, the parameters are distinguished between u- and v-direction.

In case of NWSE curve specification, the parameters are managed for each curve individually → *Poly Curve Systems*. (For further information please regard the separate manual for PCS, here the basics of this concept are handled and it contains much more details and explanations about the available options and parameters with their influence in the light deflection.)

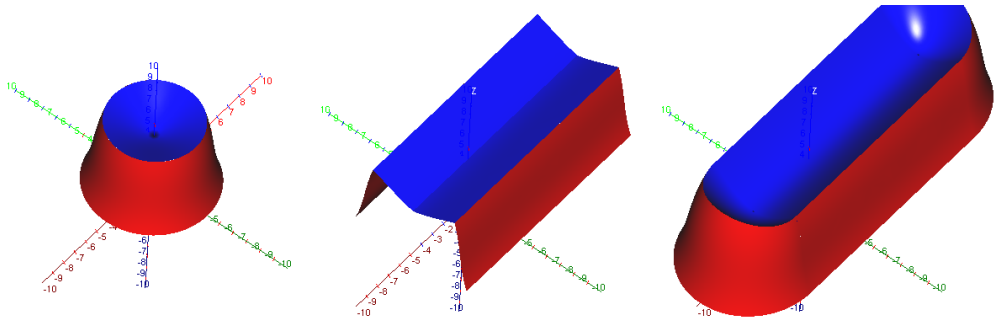


Figure 5.30: Some different extruded Center

In this figure one can see different center geometries, from the left to the right: Round symmetric, extrusion symmetric, extrusion round WNES curves. The control of the length of the extrusion part is identical in the geometries 2+3, the only difference are the two tail-pieces.

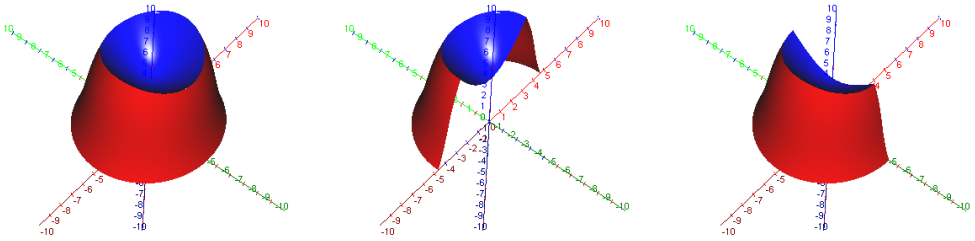
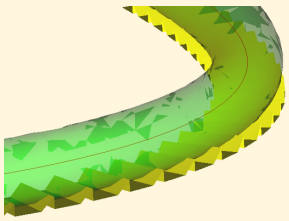


Figure 5.31: Some different extruded Center

This figure shows different center geometries created with PCS, from the left to the right: WNES curves, WNE curves, NES curves. Each given curve controls one base curve, the angular distance between two neighbouring curves is fixed at 90°, the part between them is smoothly interpolated according to the given values. Thereby, if there are only three base curves defined, only one half of the center is created (e.g. as a tail-piece for an extruded center).

5.9. Prismband for Lightguides



The prism-band function allows you to create a disturbance in 1D light-guides (light-pipes). You can use triangular shapes (prisms) or rounded arcs to carve into the backside of a light-guide, creating light spread towards the front side of the light guide. The spine curve of the light-guide is used to create the prism-band along this spine curve. The out-coupling efficiency is controlled by the distance between individual prisms and their geometric size relative to the light-guide itself.

A more detailed description about the options, parameters and the base setup can be found in a separate whitepaper, available at our website at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/whitepapers/.

Basically, the PrismBand dialog allows you to create identical, varying or individual prisms along the base curve. Therefore, all prism parameters can be defined as constant values (const), as a list of (equally spaced) control curve values, or as individual values for each prism.

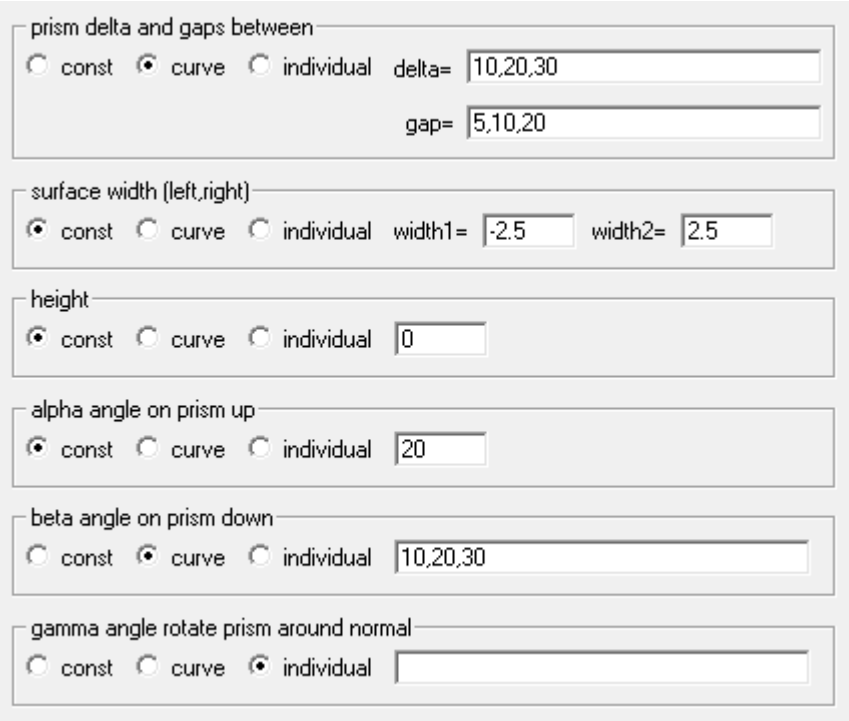


Figure 5.32: The upper part of the Prism-band Feature Dialog

radio button: prism delta and gaps between	The delta is the length of the prism along the curve, or its width if you look at the prism cross section from the side of the light-guide. It is measured from beginning to end of the prism along the spine. A value of "2" would create a prism with a length of 2 mm along the light guide. Using the option 'const' will create identical deltas and gaps. The parameters entered when using 'curve' are taken as points on a fitting curve, they always have to be entered as pairs. The last choice 'individual' uses pairs of parameters too, but the user is free to specify a certain number of prisms to create (i.e. one for each pair).
surface width (left, right)	The values of width 1 and 2 specify the width of the prisms across the light guides spine curve. Identical absolute values (e.g. -5 and +5) will create a symmetrically wide shape. Think of this width as the extrusion length of the prism cross sectional curve. (In case the option for deltas and gaps is set to 'individual', the number of pairs for all four input fields have to be the same.)
height	Used to vary the height of prism(band) above the spine curve, additional to the translation (see below). This is used to create a wedge profile e.g. for pre-collimated TIR optics.
alpha angle on prism up	The upward slope of the prism, its starting angle counted vs. the tangent of the spine curve at the starting point of the prism. This

	is the prism facet facing the starting point the spine curve. Intentionally, this angle controls the out-coupling of the light.
beta angle on prism down	The downward slope of the prism, its contact angle counted vs. the tangent of the spine curve at the end point of the prism. This is the prism facet facing the end of the spine curve. Intentionally, this angle gives additional control on the depth and the width of the prism, allowing variations such as a sharp saw-tooth profile or symmetric triangle cross section vs. an asymmetric flat end shape. (Make both angles identical to create symmetric prisms, e.g. for two-way illuminated light-guide systems.)
gamma angle rotate prism around normal	This angle rotates the prism base surface normal around the spine curve, e.g. like a helix. Usually used with the control curve option to compensate or accomplish the shape of a certain light guide design.

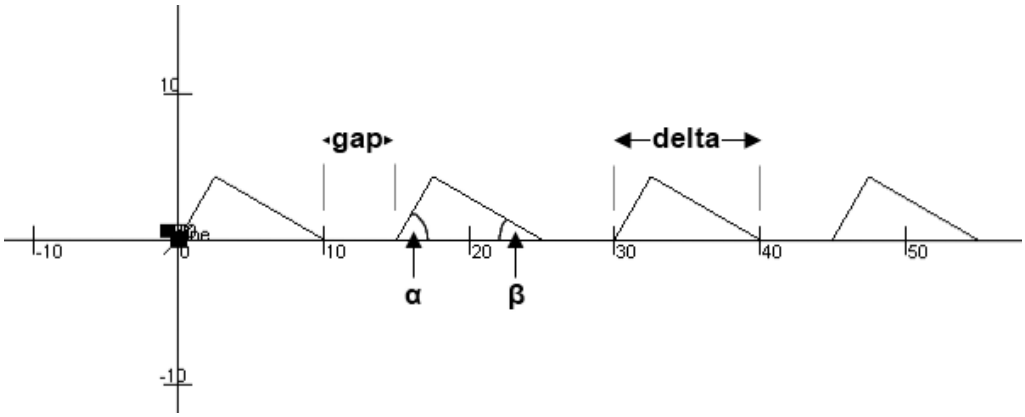


Figure 5.33: An example of the prism-band, showing some used basic parameters

This picture right above shows an example setting of the prism-band, used are only constant entries. The delta is set to 10, the gap to 5 (the delta is not defined as the length of each piece, but along the spine curve). The height is 0 (no offset from the spine curve), α is set to 60° (ascending angle of the single prisms) and β to 30° (descending angle).

This easy visually demonstration works (among other) when using the option *with base surface* (here α and β are measured against the base side of the prism triangle) and with a *light direction vector* of (0,0,1) (here α and β are measured against this vector). For a sketch of the angle calculation, please refer to the whitepaper mentioned at the beginning of this subsection.

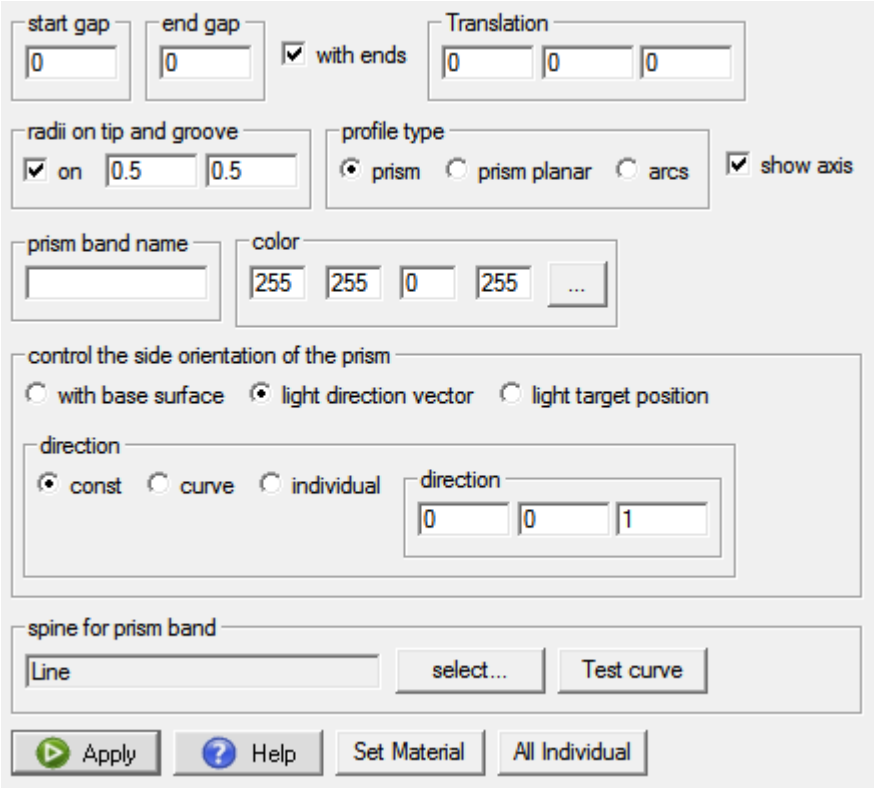


Figure 5.34: The lower part of the Prism-band Feature Dialog

start gap / end gap	Specifies the length before the first (start) or after the last prism (end) to adjust or shift the band to the actual length of the used spine curve.
flag: with ends	If checked, a connecting surface between the last prism and the end of the spine curve is created.
translation	Shifts the starting point of the prism-band relative to the global axes. This is used for example to move the prism-band to the back of a light-pipe if the center spine curve was used.
flag: radii on tip and groove	If checked, one can replace the sharp edges of the prisms with curved surfaces with the respective radii.
radio button: profile type	Switches between flat wedges of triangular cross section or arcs to create rounded instances with the specified alpha and beta angles (NURBS curve of degree 2). The only difference between the two prism types is the calculation of the start point for the next prism. Using *prism planar*, the start point for the next prism is taken without any modification, the corresponding gap is just connecting the end of the previous prism and the start of the next prism. Whereas when using *prism*, the point is calculated as the spatial centre of the end of the gap and the start of the prism. This results in a modification of both the previous gap and the following prism.

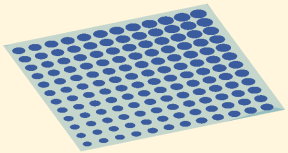
show axis	Check this box to display the axes of all prisms. This can help you adjusting the main parameters, especially gamma angles and gaps.

Finally, the desired light direction has to be specified to control the side orientation of the prisms. This has major influence on the calculation of the prism angles. Also the spine curve must be chosen.

with base surface	An additional base surface can be linked here. Each prism along the spine curve will obtain its normal from its projected position on this surface. This has certain advantages for decorative or ambient lighting designs.
light direction vector	A certain vector is defined as the desired light direction (i.e. the individual optical axis of the light to be designed). This is the standard setting for signal automotive functions.
light target position	This is the near-field setting of the light direction vector to focus light output onto a certain position. Can be also used to specify a position in global coordinates to create convergent or divergent light spread or to compensate divergence in circular light-guides.
spine for prism band	Select a curve to create the prism band on it. You also can choose to create a test curve (via the test curve button).
button: test surface/curve	Pressing a button will insert a base surface/curve for testing the different settings of this dialog.
button: set Material	If pressed, an additional dialog pops up. In here one can choose the material among various different types of either refractive or reflective behaviour and set the refraction index.
button: all individual	If pressed, a table dialog pops up. Within every single value for all parameters can be set individual. The changes can be adopted into the model via *Set* and the data can be im- or exported in the .csv file format.

5.10. Dotmask for Backlighting

The Dot Mask for back light feature creates the geometry pattern to scatter light from within a light guiding device to break the condition of total internal reflection and couple the light out of the device. Typical application examples are LCD back light plates, keypad lighting of hand-held devices or backlit acrylic plates for advertising purposes.



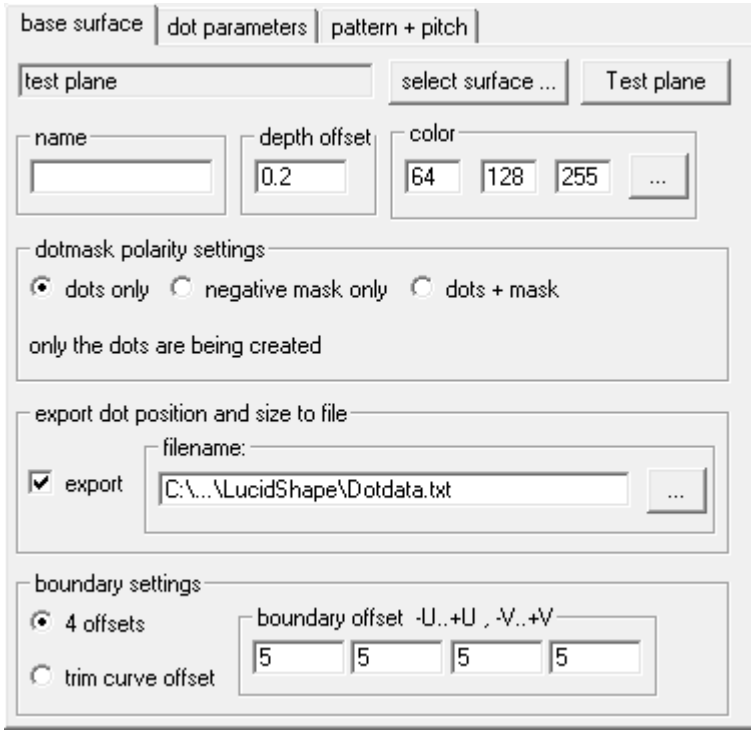


Figure 5.35: The Basic Surface Part of the Dotmask Dialog

The dot mask dialog features three separate tabs to control the base surface, the parameters for the dots, for the pattern and the pitch. The pitch is the distance between two neighbouring dots, either in u- or v-direction.

button: select surface	Opens the pop up dialog to select an already existing surface, alternatively a test plane can be used. This surface is the base surface the dots will be placed onto. (This test plane is a plain rectangular surface.)
depth offset	For a 2D dot modeling, a physical depth has to be specified. The values should be significantly lower than 1mm.
radio button: dotmask polarity settings	The respective description is shown directly below the choices
flag: export dot position and size to file	If checked, with pressing apply the complete dot setup is saved into the specified file (make sure to have writing permission in this folder).
boundary settings: 4 offsets	Sets the boundaries of the surface area covered by the dotmask, settings of 0 (zero) will make use of the full surface area. Specify here the distance to be kept from the four surface boundaries +u,-u, +v, -v. Furthermore, this option can be used to create a specially limited segment for the use of multiple dotmask instances on the same base surface.
boundary settings: trim curve offset	Specifies the boundaries in case of using a trimmed surface as the base surface.

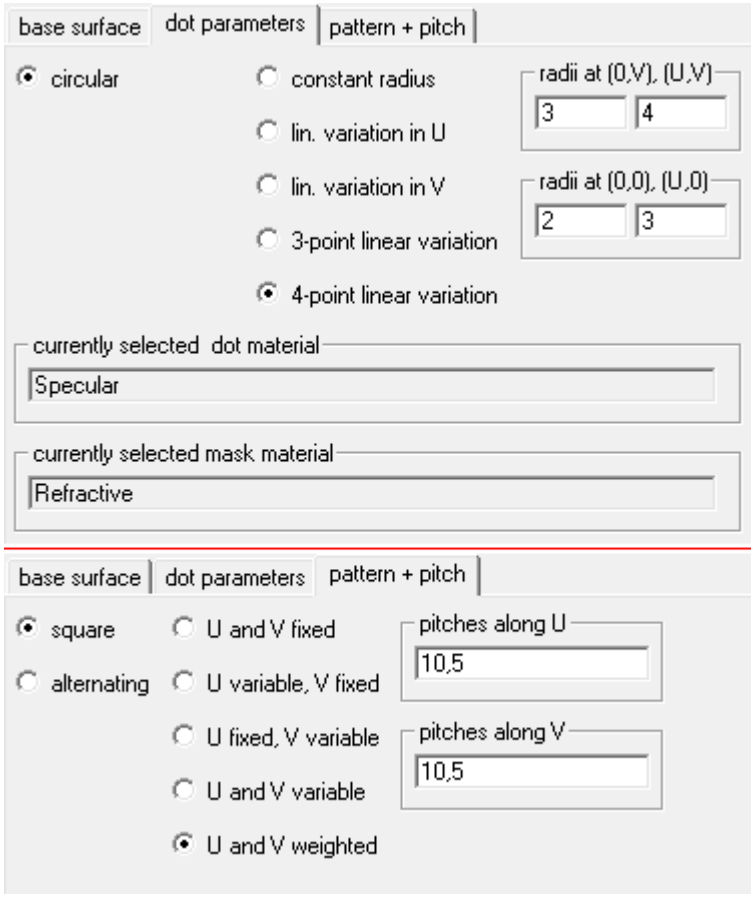
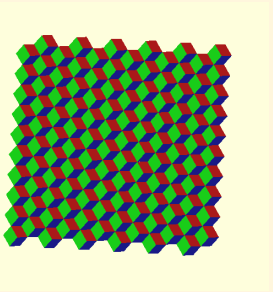


Figure 5.36: The remaining Parts of the Dotmask Dialog

radio button: radius	Determines the radius variation of the single dots, whether they are created with a constant radius or not. Available variations are a linear ascending radius (either in u or v) or a linear interpolation (either 3- or 4-point). Parameters to be entered are the respective starting and ending radius.
radio button: square/alternating	Determines whether the dots are arranged in a square array (like a chess board) or alternating (e.g. 4, 3, 4, 3, ...)
radio button: pattern + pitch	Determines the distance between two dots, either in horizontal or in vertical direction. Similar to the settings for the dot size just above, the respective distance may be set to both directions fixed, one direction fixed, both variable or weighted (manual list of values).

5.11. Retro Reflector



A common reflector is an extended unstructured surface in the three-dimensional space, it reflects the light due to the reflective material. The principle of the Retro Reflector is not based on the material of the surface, but on the spatial arrangement of the single surfaces. It uses the principle of total internal reflection to reflect the light back in the direction of the light-source. To achieve this, the surface structure is build up from small corner reflectors, each one formed by a certain number of side faces, depending on the currently selected type.

The reflected directional beam optionally can be widened. There are three base types available to choose among: Cubic geometry (build up from three side faces), twin cubic geometry and squared pyramid geometry.

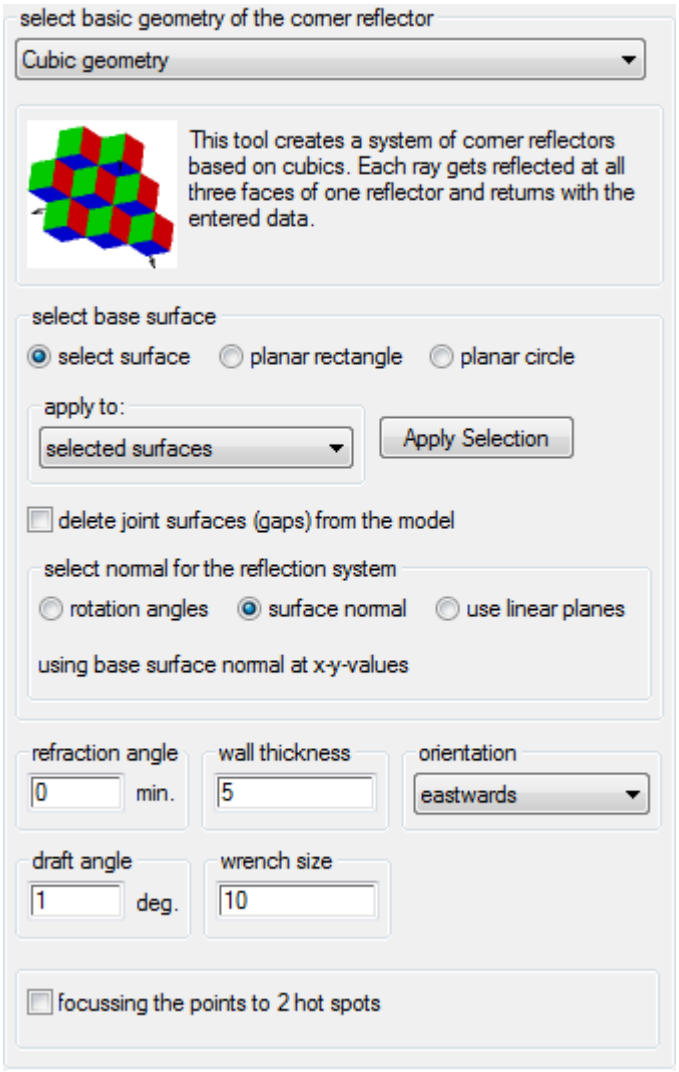


Figure 5.37: The Retro Reflector Dialog

Each single surface automatically gets a certain color assigned, depending on the function of the surface. The main surfaces are drawn in red, blue or green. The joint surfaces between the main

surfaces are shown in yellow, these additional surfaces either can be created or omitted. If necessary the yellow surfaces will be connected with magenta surfaces.

drop-down menu	One can choose among the three above mentioned base types. Each of them has slightly different options and/or parameters to adjust the behaviour.
radio button: select base surface	To create the corner reflectors, one or more base surfaces are needed to arrange the reflectors on. One can select either an already existing surface or create a new very simple planar surface which lies in the xy-plane. • select surface: In case you already have your own surface(s), choose one of the three choices in the drop-down menu (depending on how the surfaces are available in the TreeView). Next mark all the respective entries in the TreeView and press *Apply Selection*. Please note: When using *selected ObjectVectors*, the surfaces have to be in the layer directly below the selected ObjectVector (not in a layer below). • planar rectangle: This option will create a planar rectangular surface to be used as the base surface. • planar circle: This option will create a planar circular surface to be used as the base surface.
flag: show joint surfaces	If de-activated, the yellow joint surfaces completely will be removed from the model. They will neither be shown in the GeoView nor they will take part in the simulation process. This flag is available only in case of *selected surfaces*, because with planar surfaces there will be no gaps at all.
radio button: select normal for the reflection system	This choice determines the orientation of the single pyramids, more precisely the relative direction the tip points at. • rotation angles: The default orientation (0,0,1) can be altered by these three angles. When using *rotation angles* a temporary plane is created. Its normal is rotated by the given values. Each geometry is calculated to this normal. So the normal of the plane becomes 'the global normal' for all geometries. • surface normal: The normals are taken directly from the points of the base surface. • use linear planes: Here the surface is approximated or re-created by a grid of planar rectangles. This is a bit similar to the concept of tessellation, a finer graduated grid gets closer to the original surface, resulting in overlaps. The normals are taken from these assistance planes (please also see below).
# surfaces	The assistance surface is created by splitting the original base surface into the given number of planar facets. These facets can be used to determine the normals for building up the pyramids.
flag: hide planes	If checked, the assistance surface(s) will be hidden.
refraction angle	The reflected beam can be widened by choosing an angle larger than zero. The value has to be entered in angular minutes.

wall thickness	Determines the thickness, i.e. the distance between the corners of the pyramid and the base surface. In case some of the corners are breaking through the base surface, either try increasing the wall thickness or change the orientation of the pyramids.
orientation	Determines the orientation of the pyramids in the plane (more precise the starting direction of the first pyramid). This choice may have influence to the number of objects to be created. The meaning of the two directions (*north and east*) is very similar to those in the PCS application.
draft angle	In case of using a non-planar surface, the normals (used to determine the orientation of the pyramids) along the surface are not parallel to each other. The draft angle can be used to incline the pyramids' surfaces to make the production of the final product more easy.
wrench size	Determines the (wrench) size of the single pyramids, i.e. the radius of the in-circle of a pyramids floor area.
width element in ratio to wrench size	This value determines the aspect ratio of the rectangle and thus the form of the pattern. (only available for the rectangle base type)
flag: focussing the points to two hot spots	Activating this flag has to effects. The entered value for the refraction angle is overwritten and the six light reflexes are combined to two hot spots.

The default orientation of the reflector (and thus the pyramids due to the calculation process) is as follows: The emitter is located somewhere at z›o emitting the light towards the x-y-plane, so the reflector reflects the light back in +z-direction.

5.12. Genetic Algorithm Optimizer

The Genetic Algorithm Optimizer (or GAOptimizer) is a new tool for assistance in the development of lighting systems surfaces. It is automatically altering a parameter set of optical components, simulating and evaluating each experiment. The parameter space is user-defined and the evaluation is done with a pre-defined merit function. This merit function forces the optimizer e.g. to approach a uniformity criterion or to equal a certain LID. This can be done with two algorithms, either with a genetic one or via the simple system of total permutation.

The options and parameters of the Optimizer, as well as a short tutorial with an exemplary setup and description of the optimization process are explained in the separate manual *GAOptimizer.pdf* to be found on our website at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/whitepapers/.

5.13. Model Note

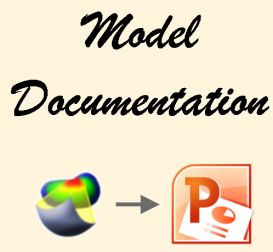
This design feature enables a text window, which then is part of the model and is saved with it. This text window is to be intended to consist e.g. of a short description or documentation when saving a row of models with different development states, to show a list of used options and parameters, etc.

After being created, the design feature is listed in the TreeView, here it can be deleted or the Edit-Dialog can be opened.

Due to the internal programming of this feature, no quotation marks (") are allowed.

5.14. Model Documentation

This tools offers a quick and easy model documentation, created as a Microsoft PowerPoint® presentation. Among other, one can choose the single objects the presentation shall contain (GeoView shaded, GeoView wireframe, UVDate, test-table, additional description, etc.), as well as the layout of it.



The model documentation can be used e.g. to record kind of a time-line during the developing process or to create a full presentation of the finished project.

Due to the internal programming of this feature, no quotation marks (") are allowed.

Please note: This design feature so far only works with the Microsoft® software in the versions 2003, 2007 and 2010. Using the 2003 version, the user has to start the software manually before creating the documentation, in later versions this initialisation is done automatically.

Basically the dialog consists of two parts: In the left part the single pages appear (analogue to the transparencies in PowerPoint). The arrows control the order of the marked page in the resulting presentation. The plus icon creates a new page, minus deletes the marked one. The ± flag determines whether the single pages shall appear in the presentation or not (this can be used e.g. for a short update, if not all pages are necessary).

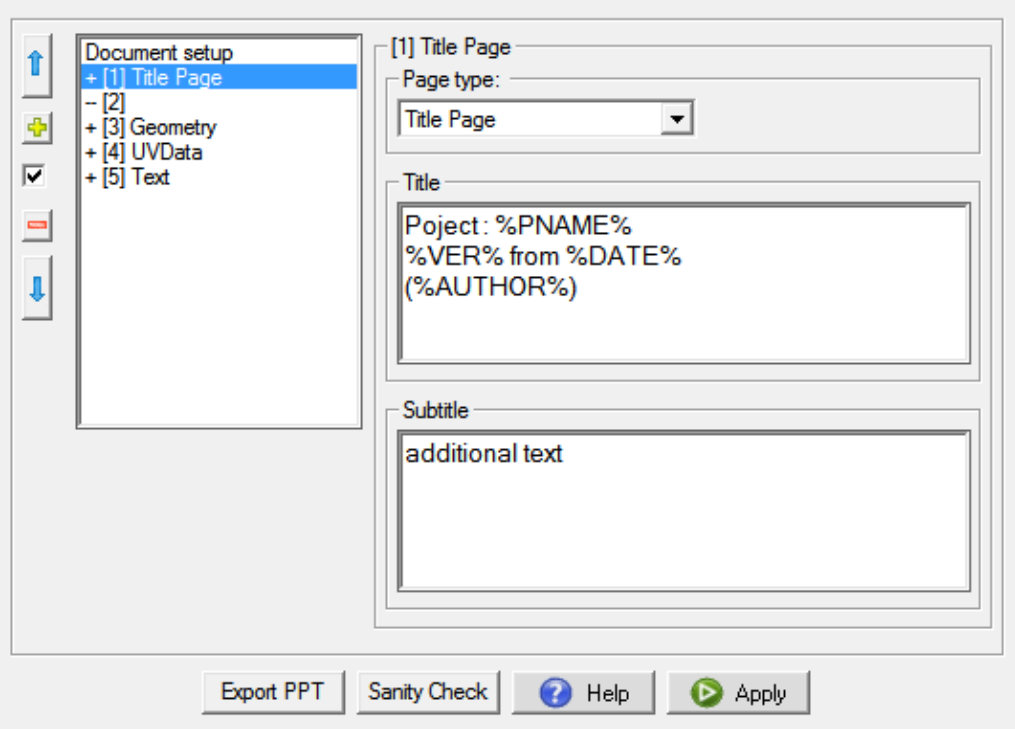


Figure 5.38: The starting Dialog of the Model Documentation

The right part of the dialog shows options and input boxes in dependency on the type of the respective page. This type can be chosen among several possibilities which all have different functions in the complete presentation.

Document Setup

% STRING%	Here four strings can be pre-defined for later use in the presentation. When creating the documentation, these placeholders will be replaced with the respective strings. Thus the respective entries only have to be changed in the setup part, not on every single page.
PowerPoint template	Here a template can be chosen, e.g. with a company logo and predefined fields and options (centered, left-/right-aligned).
button: clear template	If pressed, the template choice is cleared.
button: export PPT	If pressed, the presentation is created using the current settings.
button: sanity check	If pressed, the settings will be checked against some sources of errors.
button: apply	If pressed the current settings are applied into the DesignFeature in the model.
page title	Determines the page title to appear in the presentation.

Types of Pages

The following table lists the different page types and the available options and parameters.

blank page	Creates a blank page, e.g. to separate different parts in the presentation.
title page	Creates the title page of the document, here the pre-defined strings may be used and/or any other text.
geometry page	Selectable are single, double and quadruple GeoViews, shown are all currently visible objects. The resolution of the single GeoView and the respective background color can be set for one page, but independently to other pages. The distance between multiple GeoViews can be set via the "separate images by" value in [cm]. The visibility can be set on/off independently for all MultiSurface-Shapes, i.e. the main parts of the geometry. Single: Shows the GeoView in the specified view type (wireframe, shaded or triangles). Double: Shows the GeoView in both wireframe and shaded view. Quadruple: Shows the GeoView the same way as if using the LucidShape "4 main views" selection. Again the view type is selectable, but identical for all four views.
UVData, Driver View, Bird's Eye View, Street Luminaire View	These four page types are very similar, they all allow to add a page containing the respective feature. The headline style can be chosen among the page title, the sensor name and both together. The name neither is the name of the shape nor the material, but the name of the LIDData. The image type can be chosen among windows meta file and bitmap. The UVData to be used is selected via the pull-down menu, this list contains all existing LIDs. The properties of each page type can be set by pressing the respective button *edit ... view properties*, this always opens the usual properties dialog of this feature.
text page	Beside the page title, this page type has no options/parameters at all. It can be used to contain any comment, summary, description, etc.
test-table page	The headline style is very similar to the type 'UVData page' and the font size is self-explanatory. UVData: The upper choice determines the UVData to be used, the lower choice the TestTable. In both cases select either the corresponding object from the TreeView or a file from your computer.
list page	This type is suitable for a summary, report, table of differences, a list of things still to do, etc. Just enter a string as 'content' and choose the 'level' of this entry in the table. Available types are unnumbered and numbered lists, they cannot be mixed on one page.

external document	This last page type allows to include an already existing PowerPoint file into the documentation.

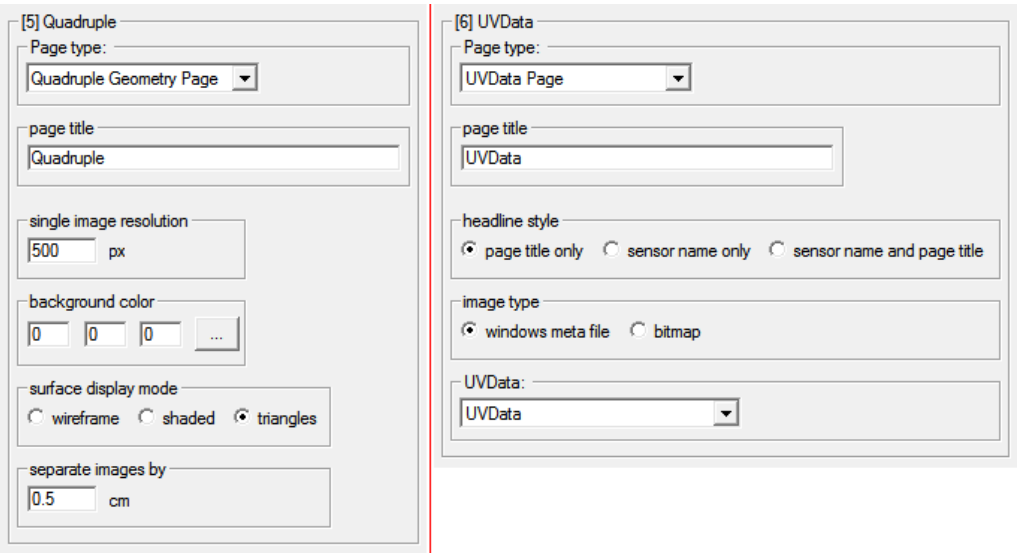


Figure 5.39: Two Exemplary Pages of the Documentation

Chapter 6. Light Sources

The light source is one of the fundamental components of every optical setup. Besides the simple point source, there is a multitude of differently shaped light sources available. This is taken into account within LucidStudio with the predefined light sources treated in this chapter.

You can choose between classical emitter shapes and special ones. Classical light sources are the cylinder, u-shaped cylinder, the typical filament coil or the geometrical point source. Special light sources are an LED point light-source, but you also can create a light source from an external ray file. The skylight source represents the sunlight illumination.

Ray bundles for ray tracing and file operations are treated in a separate section, because of their importance for the light source applications.

The emission characteristics, both spectral and spacial, can be selected for each light source in LucidStudio.

- The spectral characteristic can be defined via the Spectral Power Distribution (SPD)

The SPD can be defined by either the color temperature (e.g. ~3000K for a halogen light source) or by a step function of normalized emissivity [0,1]. This step function may be sharp or smoothly fitted and follows equidistant steps from start to stop wavelength. It is also possible to use a parameterized Gaussian spectral distribution function. If you select "no" SPD, the light will be treated geometrical only, so without a wavelength or spectral distribution. Please refer to Section 7.2, “Emitter Materials” or the Technical Reference Manual for more background on light sources and their emissions.

- The emitter type (classical) defines the spacial characteristic

The most common source is the Lambertian type. A Lambertian surface appears of constant brightness to the observer, independently of the angle of view. The $\cos^N$ type is a weighted Lambertian type, where you can choose the HWHM of a cosine function. This type is more directional than the simple Lambert type. However, you can limit the angular range of emission with both emitter types (in the respective dialogs the half angle has to be entered). Also, you can choose the directional emitter type and finally, an isoradiant emitter type. Of course, you can also import emitter characteristics from LID files.

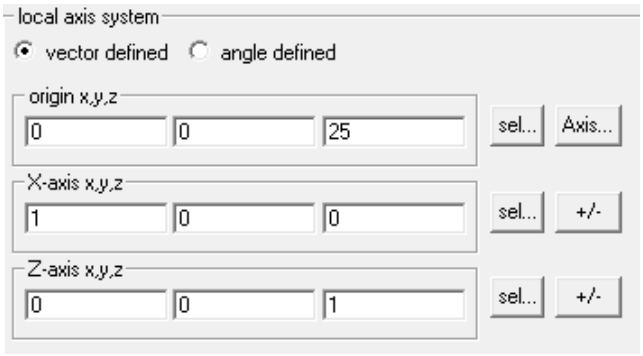
- Special light sources - specific properties.

A good example for a special light source is the LED point source. Here you can define the angular emission characteristic in detail. Thus, it is possible to simulate any possibly LED that is not yet stored in our database, but you can also evaluate the impact of altered LED properties within an existing optical setup. Another example you may be familiar with is the H4 low beam source of an automotive headlamp, which is essentially a cylindrical light source that comes with an additional absorber shape, the "shield". Some of these special light sources you will need more often than others. In any case, they provide you the same ready-to-use shape every time for reoccurring applications.

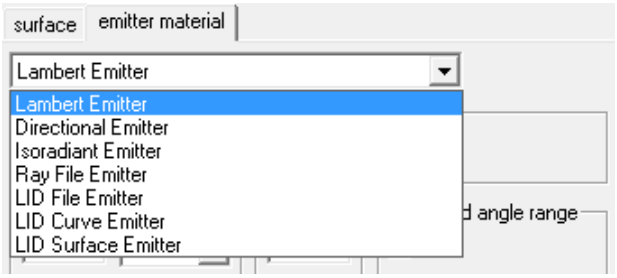
6.1. Base Options and Parameters

This sections describes some of the options and parameters which are common to (almost) all light-sources and their respective dialogs.

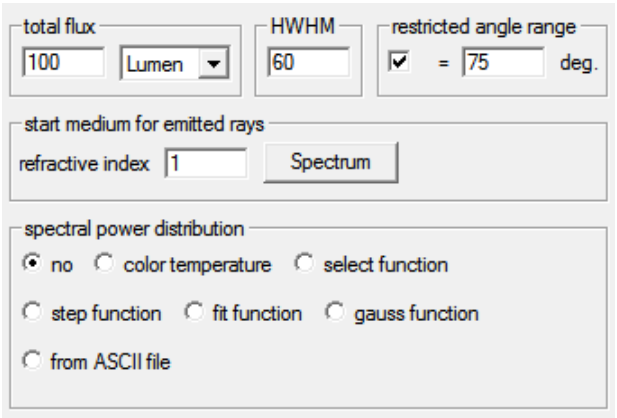
- Local axis system to specify their location and orientation (explained in detail in Section 4.10.1, “Create Axis System ”).



- Specification of the light source emitters.



This determines the base characteristic of the emitted light. This e.g. can be directional (all emitted rays are parallel to each other as long as they do not intersect), isoradiant (isotropic spacial emittance) or something intermediate (explained in detail in Section 7.2, “Emitter Materials”).



total flux	Sets the total output of this light-source to either [lm] or [W].
half width at half max (HWHM)	Measured in the profile of the beam, the value determines the distance between the centre of the outgoing light and the point at which the power is down to 50% in the profile, this gives the spatial distribution of the outgoing light. (E.g. a very small value refers to a almost laser-like beam, whereas a very high value means almost parallel light.) Please also see the sketch below.
flag: restricted angle range	If checked the solid angle of the emitted light can be restricted. The entered value sets the maximum allowed angle relative to the normal. Within this solid angle the light intensity is distributed as specified by the HWHM value.
start medium for emitted rays	The refraction index (n) of a starting ray can be set manually. This can be used if the source shall be inside a medium, e.g. in a light-guide. (The default value is n=1 which means a ray starting point in air.)
radio button: spectral power distribution (SPD)	The SPD controls the behaviour of each ray depending on its wavelength. For a detailed description, please refer to Section 7.2.8, “Spectral Power Distribution (SPD)” or Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions”.
efficacy	Determines the efficacy of the lamp. In the complete final system there are losses of light (in the lens, the reflector, air, etc.), with this parameter this global loss can be taken into account to the lamp itself. The range of the parameter is the interval [0,1]. Simple example: When using an emitter with a flux of 1lm and a reflector with a reflectance of 0.8, there is a loss of 20% → Entering an efficacy of 0.8 scales this emitter in a way that the resulting flux in the sensor is 1lm again. (Please refer also to Section D.1, “Efficacy Scale Factor”)
flag: emitter in inverse direction of surface normal	If checked the created light-source emits the light in the opposite direction relative to the surface normal ("illuminates backwards"). This can be helpful e.g. to correct a mirrored parametrization of a shape.
name and color	Name determines the string shown e.g. in the TreeView. The color determines the object's appearance in the GeoView, given in the four RGB values. This has no effect onto any simulation process!

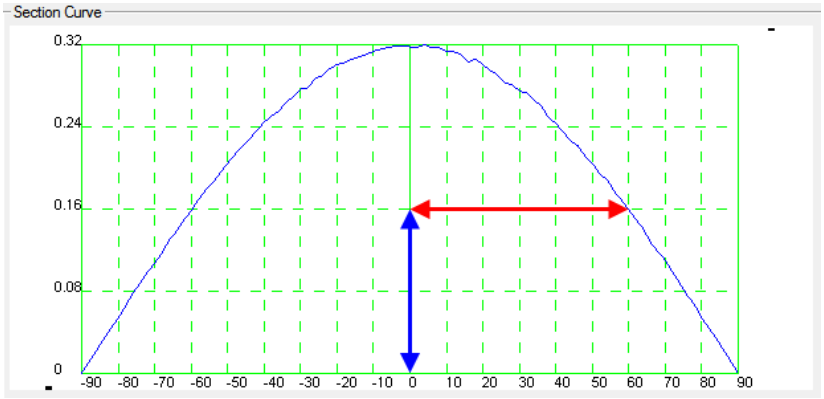
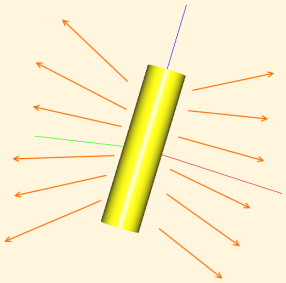


Figure 6.1: An exemplary sketch showing a HWHM of 60: The red arrow is the angular distance (this is the half width) where the curve is down to half of its maximum (blue arrow).

6.2. Classic Light Sources

6.2.1. Cylinder Light Source

This function creates a cylinder surface in combination with a light emitter material. As per default the cylinder is created symmetrical to the z-axis.



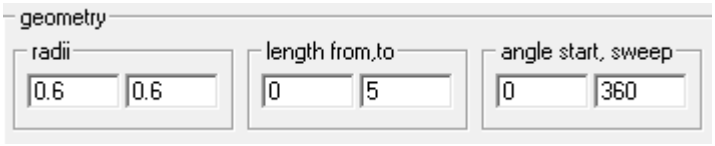
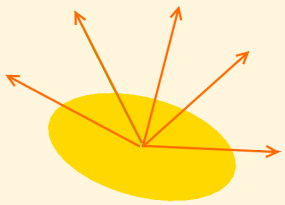


Figure 6.2: Details of the Cylinders Light Source Dialog

length	The length of the cylinder.
radii	The two radii of the cylinder. Using two identical values creates a circle as base curve, otherwise an ellipse is created.

6.2.2. Disk Light Source

This function creates a disk surface in combination with a light emitter material. Partial disks and rings may be created by specifying an angular range different from [0,360] and a nonzero inner radius.



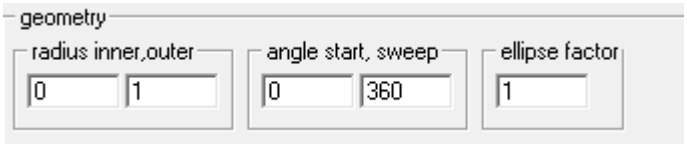
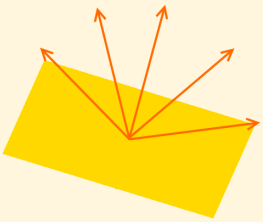


Figure 6.3: Details of the Disk Light Source Dialog

angle range start, sweep	The angular range in degrees, [0,360] is the full range.
inner, outer radius	The inner and outer radius, an inner radius > 0 creates a ring.
ellipse factor	If set to 1, a circle will be created as the base curve, otherwise an ellipse with the given ratio is created.

6.2.3. Plane Light Source

This function creates a planar surface in combination with a light emitter material.



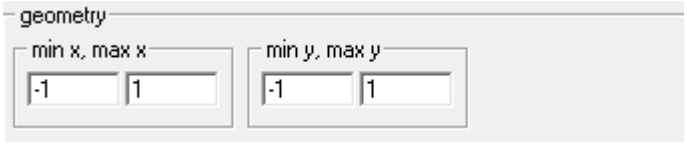
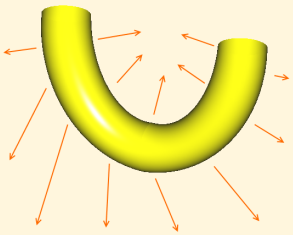


Figure 6.4: Details of the Plane Light Source Dialog

min x, max x	The horizontal range of the plane.
min y, max y	The vertical range of the plane.

6.2.4. U-Shape Light Source

This function creates an u-shaped light source. In total this geometry uses three radii, one to define the profile radius of the circle (cross section of the torus) and two to define the torus or u-shape. The two sweep angles define the start (and thus the orientation) and the end of the torus.



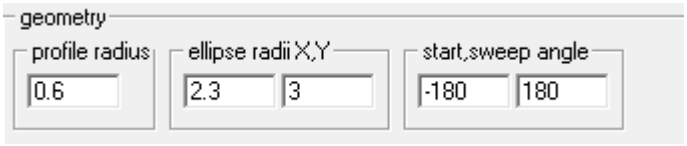
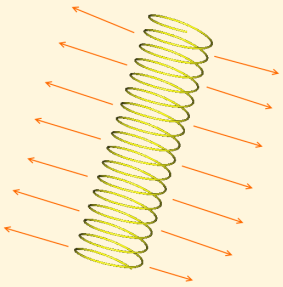


Figure 6.5: Details of the U-Shape Light Source Dialog

profile radius	The radius of the cross section.
ellipse x,y radii	The two ellipse radii.
start, sweep angle	The start and sweep angle define the orientation and a sub arc of the ellipse.

6.2.5. Coil Shape Light Source

This function creates a coil shaped light source by using a helix as a base curve. This type can be used to create the filaments of special lamp models.



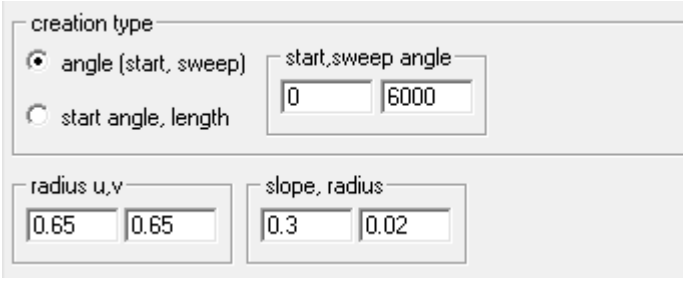


Figure 6.6: Details of the Coil Shape Light Source Dialog

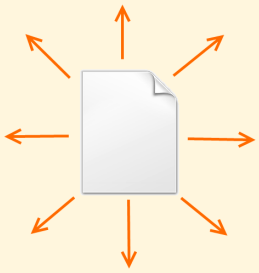
start, length / sweep angle	The start angle defines the orientation of the helix. The length of it can be determined directly via the length or indirectly via the sweep angle (i.e. the number of windings, this operates in combination with the slope).
radius u,v	The two radii of the base helix curve. If u=v, the coil has the form of a circle.
slope, radius	The slope determines the ascending angle of the helix (a higher value increases the height of the resulting coil, but not the length of it). The radius determines the inner radius, i.e. the diameter of the circle when looking from above onto the coil.

6.2.6. RayFile Light Source

Creates a light source from a ray file. When simulating this light source, only the rays in the files are produced one after another.

A ray file can be achieved by recording the light emitted from a exact model of a light-source. After that, the ray file can be used as a replacement for the emitter's geometry.

Please also refer to Section 6.6, “RayFile and RayFile Tools”.



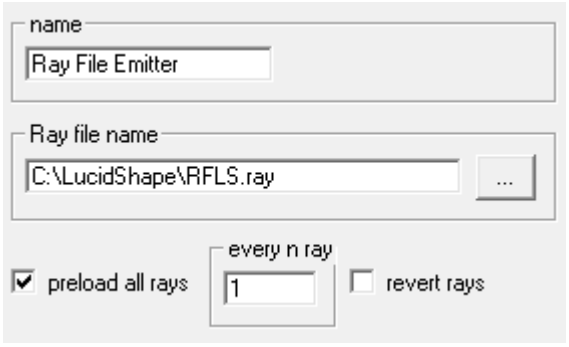


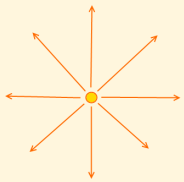
Figure 6.7: Details of the RayFile Light Source Dialog

position	Determines the coordinates and the axis of the ray file emitter to be created.
ray file name	The full path and the name of the ray file have to be entered.
flag: preload all rays	If checked, the complete ray file (i.e. all rays in it) is pre-loaded into the RAM in the moment the light-source is created or the model is loaded into LucidShape. If unchecked, the rays will be read one after another during the simulation process and deleted after use. The second choice is slightly slower but will take much less RAM (may be important on PCs with limited RAM).
every n. ray	This integer determines whether all rays (1) or every n. ray (N›1) is taken for the simulation (the other rays between will be skipped.
revert rays	Reverts all rays' directions.

6.2.7. Point Light Source

The point light source is set up by an axis system and an emitter material. The axis defines the position and orientation (z-axis) of the point source. Due to the fact that this source is one-dimensional and has no 'real' surface, the z-axis will be taken as the surface normal instead.

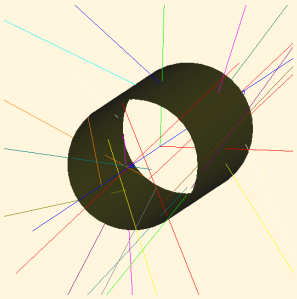
This light-source has no new options or parameters.



6.3. Special Light Sources

6.3.1. Volume Light Source

A volume light source creates random rays within a volume using a predefined directionality. This can be a cylinder, a sphere or a cone. This way e.g. the light arcs in discharge lamps like D1R can be simulated more realistically.



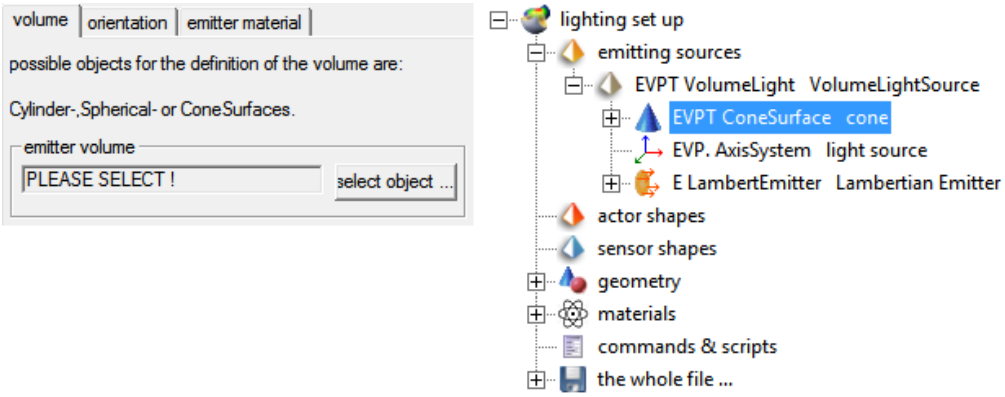
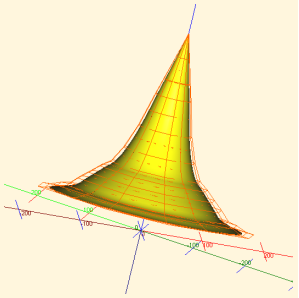


Figure 6.8: The Volume Light Source Dialog

tab: volume	Pressing 'select object' allows to select a geometry from the GeoView or TreeView, this base surface serves as the outer boundary of the volume to be created. Only cylinder, sphere or cone shaped surfaces are allowed here.
tab: orientation	Please note: Unlike almost all other light-sources, the volume light-source cannot be transformed by this tab. To alter the origin and the orientation of the emitter, open the Edit-Dialog of the marked entry and change the values within.

6.3.2. LED Point Light Source

This point light source uses a light emitter with a light intensity distribution taken from a curve. The curve is calculated as an ExplicitCurve, where the entered pairs of degree and degree value define the ValueRow and thus the directional characteristic.



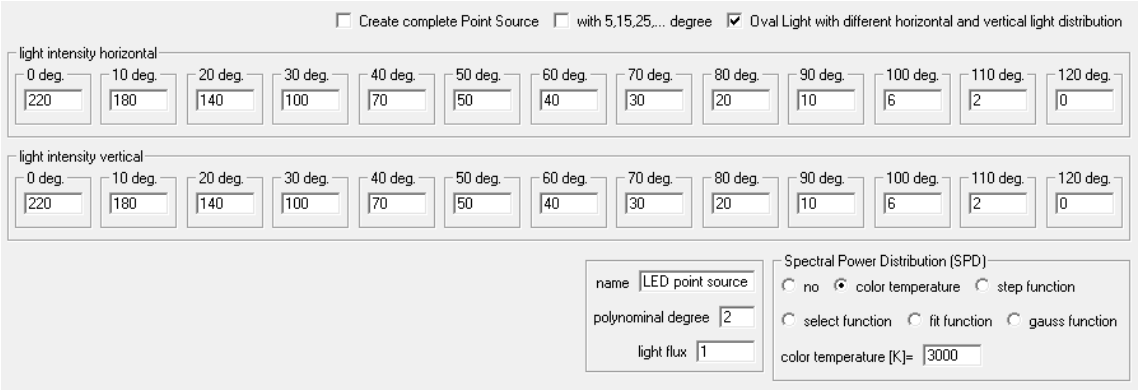
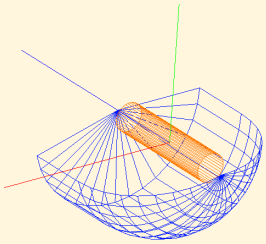


Figure 6.9: The LED Point Light Source Dialog

flag: create complete point source	If checked a complete point light source is created when pressing apply. Otherwise only an emitter material is created.
flag: with 5, 15, 25, ... degree	If checked the interim values can be entered too, hereby the characteristic can be specified more detailed.
flag: oval light	If checked it is possible to determine the vertical values to create a non isotropic characteristic.
light intensity horizontal	The entered values for 0°, (5°,) 10°, (15°,) 20°, ... define the horizontal light intensity distribution of the LED for the given angles. If the flag "oval light" is unchecked, these values are used to create an isotropic characteristic.
light intensity vertical	The entered values for 0°, (5°,) 10°, (15°,) 20°, ... define the vertical light intensity distribution of the LED for the given angles.
polynomial degree	The polynomial degree for the explicit curve built for the emitter characteristics.
flux	Determines the total light flux of the light source.

6.3.3. Low Beam, High Beam, Signal Lamp and special Light Sources

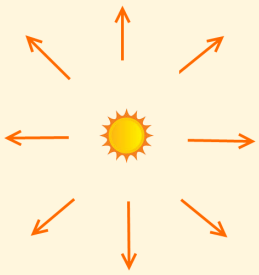
The tools in this subsection create special cylinder light sources for low beam, high beam and signal lamp applications. The specific geometry of each type is taken into account, while the basic properties are those found in Section 6.2.1, “Cylinder Light Source”. The subsequent table lists the available types.



low beam sources	H1, H3, H4 low+shield, Hs1 low+shield, H7, H8, H10, H11, H12, H13 low, H16, HB4(9006), HB5(9007) low, S1 minor + shield, S2 minor + shield, S3, D1R
high beam sources	H4 high, Hs1 high, H9, H13 High, HB3 (9005), HIR1 (9011), HIR2 (9012), HB5(9007) high, S1 major, S2 major, H15 high, H15 high rotated by 180°
signal lamp sources	P13W, P19W, P24W, PY24W, HP16W, P21W , P21W up, P21W5 major, P21W5 major, P21W5 minor, P21W5 minor, P27W7 major, P27W7 major, P27W7 minor, P27W7 minor, H21W, H21W up, R5W, R5W up, R5W U-Shape, R5W U-Shape, R10W, R10W up, R10W U-Shape, R10W U-Shape, W5W, W5W up, W3W, W3W up, W16W, W16W up, W21W, W21W up, H15 DRL
other special sources	Hs3, Hs4, QT12

6.4. Sky Light Source

Creates a light source for sky light simulation. This is useful in the headlamp design process to consider the reflection or focussing of sun light on the headlamps reflective surfaces or optics in order to prevent melting or getting damaged (e.g. cover lens, front glass, etc.).



The properties of the sky light source are determined by the geographical position on earth (longitudinal and latitudinal degrees) and time of day, i.e. the local time (hour) and time zone (meridian). You can choose between two models of the sky light source. First, there is the sun disc model, which represents the sun (2D angular width of 0.53 deg). Second, there is the clear sky model, which features a clear blue sky without direct contribution from the sun disc. Both are CIE models that are well-known in literature on solar lighting.

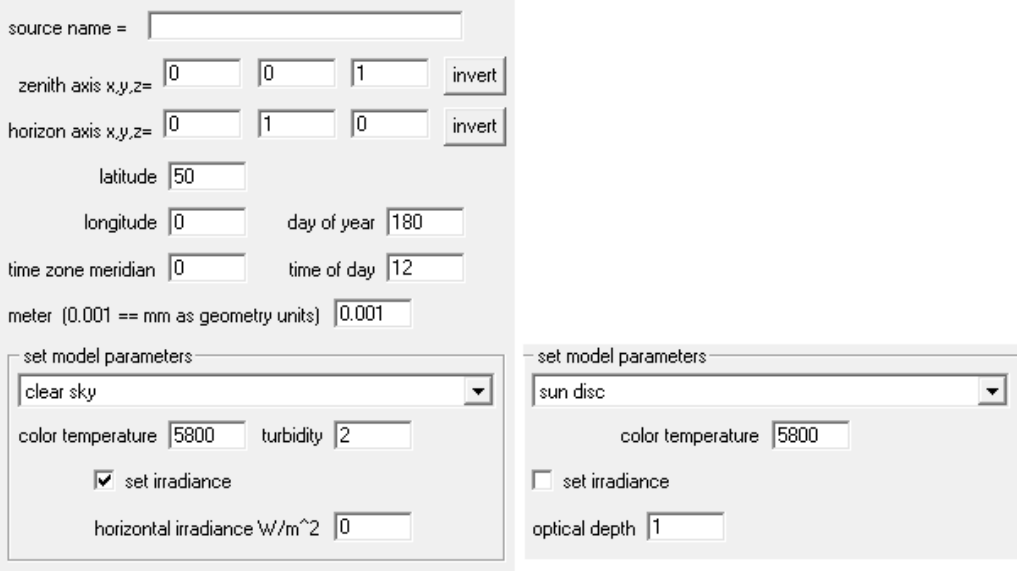

source name =
zenith axis x,y,z= 0 0 1 invert
horizon axis x,y,z= 0 1 0 invert
latitude 50
longitude 0 day of year 180
time zone meridian 0 time of day 12
meter (0.001 == mm as geometry units) 0.001
set model parameters
clear sky
color temperature 5800 turbidity 2
set irradiance
horizontal irradiance W/m^2 0
set model parameters
sun disc
color temperature 5800
set irradiance
optical depth 1

Figure 6.10: The Sky Light Source Dialog

zenith axis x,y,z	The light sources zenith direction. This defines the vertical axis of the light source model (pointing up into the sky). It may differ from (0,0,1) if your system is tilted (e.g. the headlamp of a car located on a hill side).
horizon axis x,y,z	The light source's horizon direction. Defines the lateral orientation of your axis system. If west is along (0,1,0), north will be (1,0,0).
button: invert	Inverts the respective axis, i.e. toggles the algebraic sign of the values (from positive to negative and vice versa).
latitude	Defines the geographic position (degree of latitude) of the experiment, north or south of the equator [-90,90].
longitude	Defines the geographic position (degree of longitude) of the experiment, east or west of the prime meridian [-180,180].
time zone meridian	Defines the time zone in which you specify the local time. A time zone of zero equals GMT.
day of year	Defines the day of the year, running from 0 to 366.
time of the day	The local time of day in hours, running from 0 to 24.
select: set model parameters	Select either "sun disc" model or "clear sky" model and choose the color temperature. Alternatively you can manually define the irradiance for both models. An irradiance of zero uses the results from the selected model.
flag: set irradiance	If checked the default irradiance value of the model is overwritten with the entered value.
optical depth	Defines the extinction coefficient in the Rayleigh scattering formula. It expresses the quantity of light removed from a beam by scattering or absorption during its path through a medium, here the atmosphere.
turbidity	This factor defines the haze or cloudiness of the sky (a measure of air pollution, dust, and fog).

6.5. Ray Trace Bundles

This section describes the various ray trace bundles you can choose for the interactive ray tracing (Section 1.3.5, “Simulate the scene”, the section called “Light Screen View”). LucidStudio offer specific sets of rays for the geometry view, so you can choose the one suited best for your needs. For most precise survey of your experiment, you may want to use a single ray, while spread is viewed best with larger bundles. Also, the origin or alignment of the ray bundles can be chosen, e.g. a Cylinder Filament bundle or a Parallel bundle of rays. Independent of the exact ray bundle type, you have to assign a set of coordinates to define the origin of the rays - the basic (x,y,z) position.

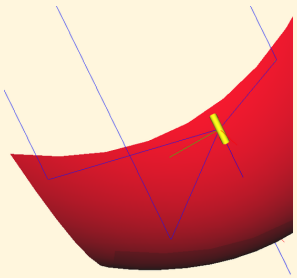
Some dialog parts are common to most dialogs in this section. Instead of repeating these parts for every tool, we describe them once here:

Figure 6.11: Common Ray Trace Bundle Parameters

name	The ray bundle's identifier.
offset	The ray bundle's offset (x,y,z), relative to the light-source's position.
color	The ray bundle's color as displayed in the geometry view.

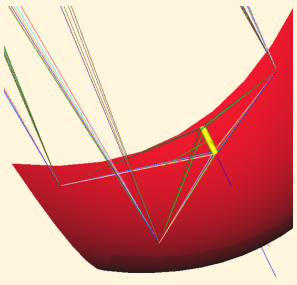
6.5.1. Single Ray Bundle

Creates a single ray bundle. The created ray bundle starts from a given position defined by the light-source's position and an optional offset to it. The origin of single ray "bundles" is always a point, e.g. the center position or a specific corner of a cylinder light-source.



6.5.2. Cylinder Filament Bundle

Creates a cylinder filament bundle. The ray bundles start from a cylinder's front boundary. The cylinder length, radius and number of rays can be entered. The position and orientation of the cylinder is defined by an offset (relative to the light-source's position) and direction.



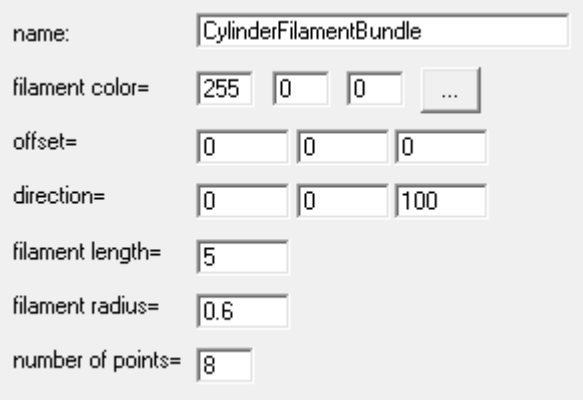
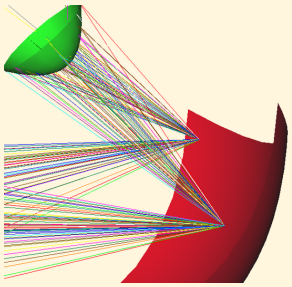


Figure 6.12: The Cylinder Filament Bundle Dialog

direction	Direction of the filament orientation.
filament length	The length of the cylinder.
filament radius	The radius of the cylinder.
number of points	The number of rays for each side of the cylinder.

6.5.3. Free Surface Filament Bundle

Creates a free surface filament bundle. The ray bundles may originate from any surface you like. On a selected surface, a point grid is created and used in a RayPathFilamendBundle object. The grid size and resolution (effectively the number of rays) can be selected via the number of points in u, v.



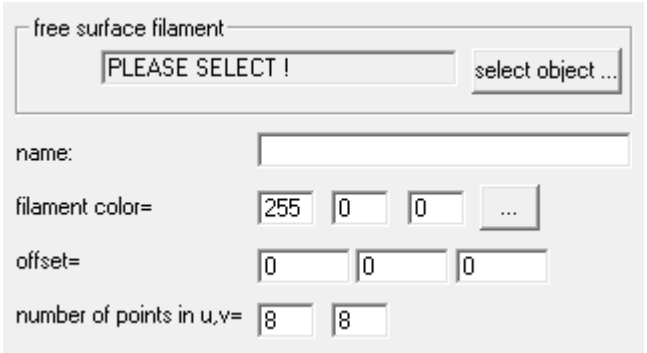
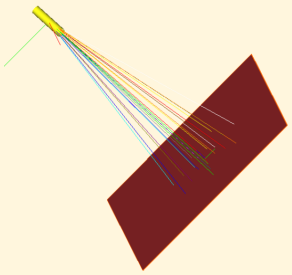


Figure 6.13: The Free Surface Filament Bundle Dialog

free surface filament	Mark any surface in either the GeoView or the TreeView.
number of points in u,v	The number of grid points in u- and v- direction on the surface.

6.5.4. Cone Bundle

Creates a cone bundle emerging from the defined source/target point. The cone angle (the angular spread of the bundle) and the number of rays can be chosen. The cone is divided into a polar grid with $n_\theta \cdot n_\varphi + 1$ rays. The orientation of the grid can be determined with the "base vector".



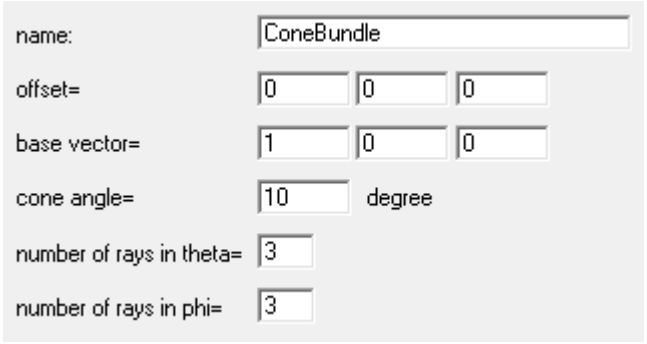
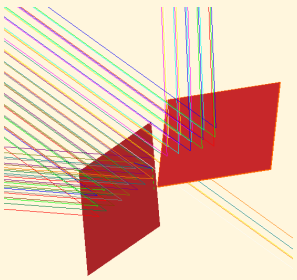


Figure 6.14: The Cone Bundle Dialog

base vector	Base vector for the polar grid.
cone angle	The cone opening angle in degree.
number of rays in θ / φ	The polar grid size which builds the cone bundle.

6.5.5. Parallel Bundle

Creates a parallel bundle. A rectangular grid of parallel rays passes the area around the target point. The number of rays in u and v, the direction of the ray bundle and the orientation of the grid can be chosen. The grid is set up by $((n \cdot u-1) \cdot 2+1) \cdot ((n \cdot v-1) \cdot 2+1)$ rays.



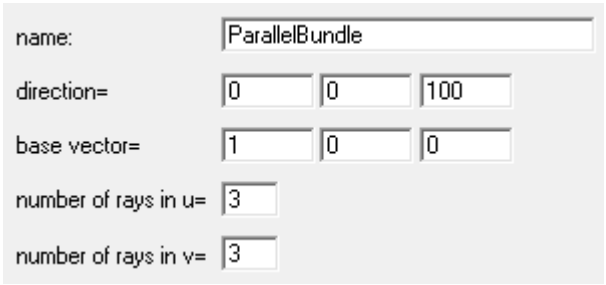


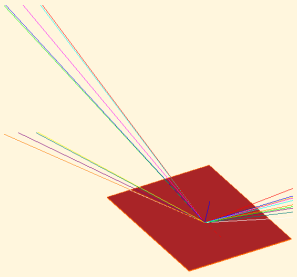
Figure 6.15: The Parallel Bundle Dialog

direction	Direction vector for all rays.
base vector	The base vector for the grid.
number of rays in u /v	The grid size.

6.5.6. Point Set Bundle

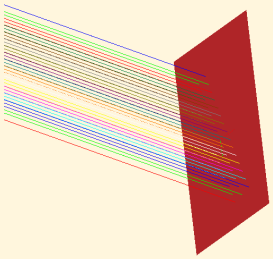
Creates a point set bundle: The rays start from a fixed set of points.

Note that the set of points cannot be edited in the dialog (which contains only the object identifier as its only editable parameter) directly. The point sets have to be created by script, then they are selectable via the "point set bundle" dialog.



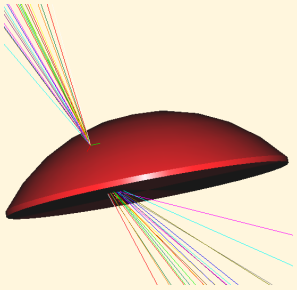
6.5.7. Parallel On Normal Bundle

Creates a parallel bundle. All rays are going to the target point parallel to the surface normal. The number of rays in u and v can be entered. The grid is set up by ((n·u-1)·2+1) · ((n·v-1)·2+1) rays. The length of each start ray and the distance between the rays in u and v direction can be entered. The flag "into back side" serves as a direction indicator and allows to switch between both sides of the surface.



6.5.8. Tip Cone On Normal Bundle

Creates a tip cone ray bundle on normal, that is to say a ray bundle with a cone of rays around the surface normal where the tip of the cone is created on the touch point. The length of the start ray, the cone angle and the number of rays in θ and φ can be entered. It can be determined in which direction (X, Y or Z) the rays should run and an indicator allows to switch between both sides of the surface.



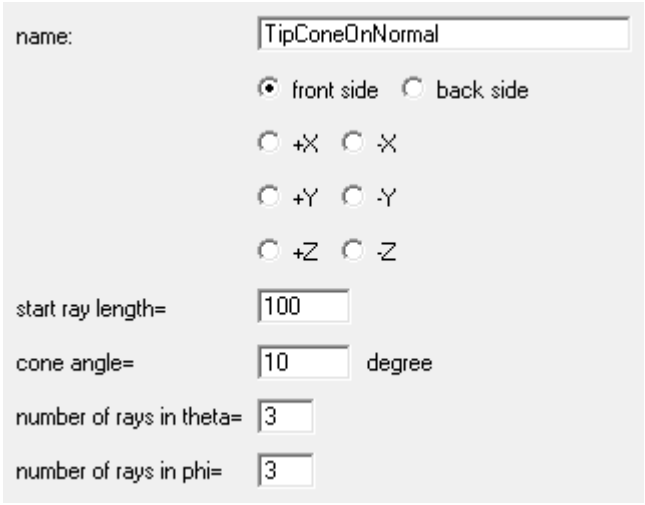
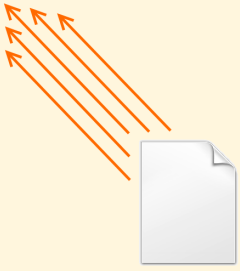


Figure 6.16: The Tip Cone On Normal Bundle Dialog

radio button: front side/back side	A direction indicator for rays going into the front or back side. If checked, this disables the +X,-X,+Y,-Y,+Z,-Z options.
+X,-X,+Y,-Y,+Z,-Z	If checked, forces the rays to propagate into the predefined direction, i.e. +X,-X,+Y,-Y,+Z,-Z. Disables the front/back side choice.
start ray length	The length of each start ray.
cone angle	The opening angle of the cone.
number of rays in θ / φ	The polar grid size which builds the cone bundle.

6.5.9. RayFile Bundle

Creates a ray file bundle. The rays are read from a file. Only those rays which are close enough to the ideal ray (from the start to the target point) are used. The angle limit "angle threshold" controls the closeness, i.e. which rays are used.



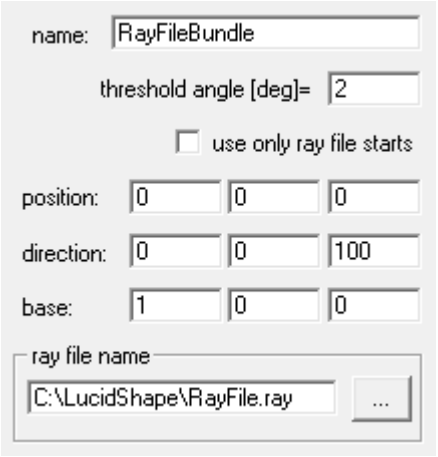


Figure 6.17: The RayFile Bundle Dialog

angle threshold	The opening angle of the rays. A smaller angle threshold leads to a narrower ray bundle.
ray file name	The path and the name of the ray file to be used.

6.6. RayFile and RayFile Tools

In this section we try to give an introduction in the use of ray files concerning a simulation in LucidShape. There are two alternatives to obtain a ray file, either by download from the homepage of the LED supplier or to record it directly in LucidShape (for a description of how to create an own ray file, please refer to Section 7.4.2, “Ray File Sensor”).

When performing a simulation, using the ray file emitter type is a much better choice than using e.g. a lambertian emitter. A ray file contains all (!) informations of the emitter, e.g. in case of a LED it contains the whole geometry of the chip, the coating and any optics like lenses.

In contrast to the other emitter types, the flux of a ray file emitter cannot be set directly in the dialog, because it is saved in the ray file. Sometimes it might be necessary to alter the flux, e.g. if the ray file's flux does not match the flux required in the simulation. LucidShape provides tools to analyse and modify ray files. Among other there are *Check Ray File* (examines the ray file and states the flux and the number of rays stored) and *Copy Ray File* (can be used to alter the flux and the number of rays). Both tools will be explained in the following sub-sections.

6.6.1. The use of a RayFile

The informations stored in a ray file consist of single rays. For each ray there are the origin, the direction and the flux. (Optional data would be spectral informations.) In a simulation the stored rays are reproduced and traced through the geometry.

Because all informations about the rays are already stored in the ray file, it is strongly recommended to assign a ray file emitter material to a point only (i.e. to create a point light source). What happens when assigning a ray file onto a surface? It creates random points and normals on the surface, additional it transforms the next random ray's positions and direction. So the stored informations about the origin of each ray are more less strongly blurred.

6.6.2. Simulation with more than one RayFile

In total there are four possibilities when using more than one ray file in a simulation:

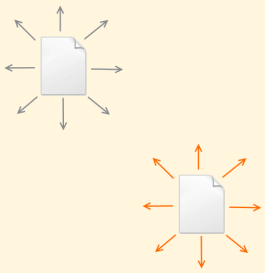
- same number of rays and identical flux
- same number of rays, different flux
- different number of rays, same flux
- different number of rays and different flux

In the cases two to four, the flux of the ray files (more precise the flux of the single rays) automatically is adjusted. This ensures both that the ray-trace process can make use of the whole number of rays stored in each ray file and that the resulting flux is the sum of the original fluxes saved in the ray files.

The RayFile format used in LucidShape is explained in detail in the Technical Reference, section 3.4. (This .pdf file can be found either in the documentation folder of your LucidShape installation (..\doc\LucidTechnical.pdf) or on our website at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/manuals/.)

6.6.3. Copy RayFile

Copies a ray file into the same or another format. You may choose either LucidShape ASCII or LucidShape binary.



input ray file name	The path and the name of the input ray file.
output ray file name	The path and the name of the output ray file.
target file format	The file format of the output ray file, available are spectral (LucidShape ASCII or binary) and non-spectral (LucidShape, either ASCII or binary).
overwrite flux	The flux of the output ray-file may be altered to this value in [lm], this is independent to the number of rays stored in it. This value is also shown in the message box for the total flux or in the text edit view for the flux of individual rays. (This is useful e.g. while using ray files with different fluxes, so one can kind of equalize the ray files.)
copy only rays with min % I	Copies only rays with a certain minimum percentage in intensity.
copy each N. ray	Copies only each N. ray, this function can be used to reduce the number of rays in the output file (e.g. to have a small ray file for a very first sketch of changes in the model).
flag: reverse rays	Inverts the direction of the rays, e.g. (0.6,0,-0.8) → (-0.6,0,0.8).

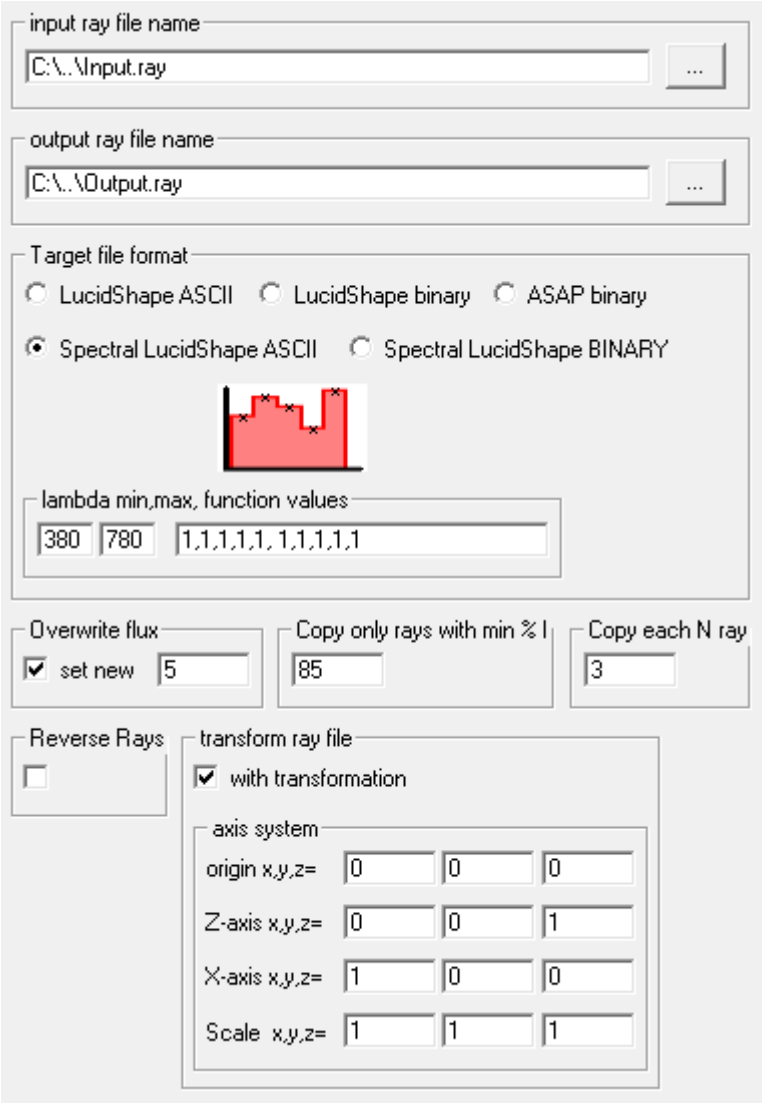
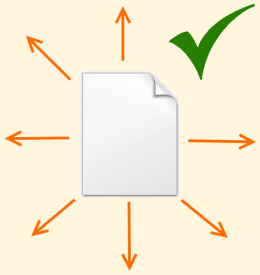


Figure 6.18: The Copy RayFile Dialog

flag: with transformation	If checked one can alter the position and the scaling of the ray file. The scaling changes the size of the emitting volume, this can be set independently for the three axis. (This procedure is analogue the scaling which can be done with the 'Transform' dialog.)

6.6.4. Check RayFile

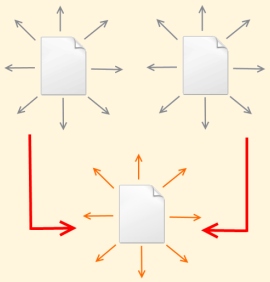
You may check an existing ray file. Pressing apply shows the file format, the bounding box, the total flux and the number of rays.



input ray file name	The path and the name of the ray file to be checked.

6.6.5. Merge RayFiles

Two or more ray files can be merged to a single one. This can be used e.g. to combine spectral ray files when having obtained a 'blue' and a 'red' one. The resulting ray file has the same format as the first in the list.



weighting factor	In case the single source ray files shall not be taken over equally into the new merged ray file, the user can weight them by entering different weighting factors for each source file. E.g. when merging four ray files, a weighting like 4:1:3:2 is possible. The weighting factors do not effect the number of rays taken over from the source ray files, but the flux in them.
flag: read step function from file	The SPD of the single ray files can be determined either by entering the values into the dialog or by selecting an ASCII file containing this data (e.g. a file obtained from the LED supplier).
flag: force export as spectral ray file	Allows to transform a non-spectral ray file into a spectral one (saved either as LucidShape ASCII or binary), using the specified SPD.
output ray file name	The path and the name of the ray file which contains the merged ray data.
button: add file	Opens a file selection box in order to select a ray file.
button: remove file	Removes the selected file from the list.
button: change file	Opens the file selection box to change the marked ray file.
button: clear list	Clears all entries in the list so that a new list can created.
button: apply	If pressed, the listed ray files are merged, using the entered options and parameters.

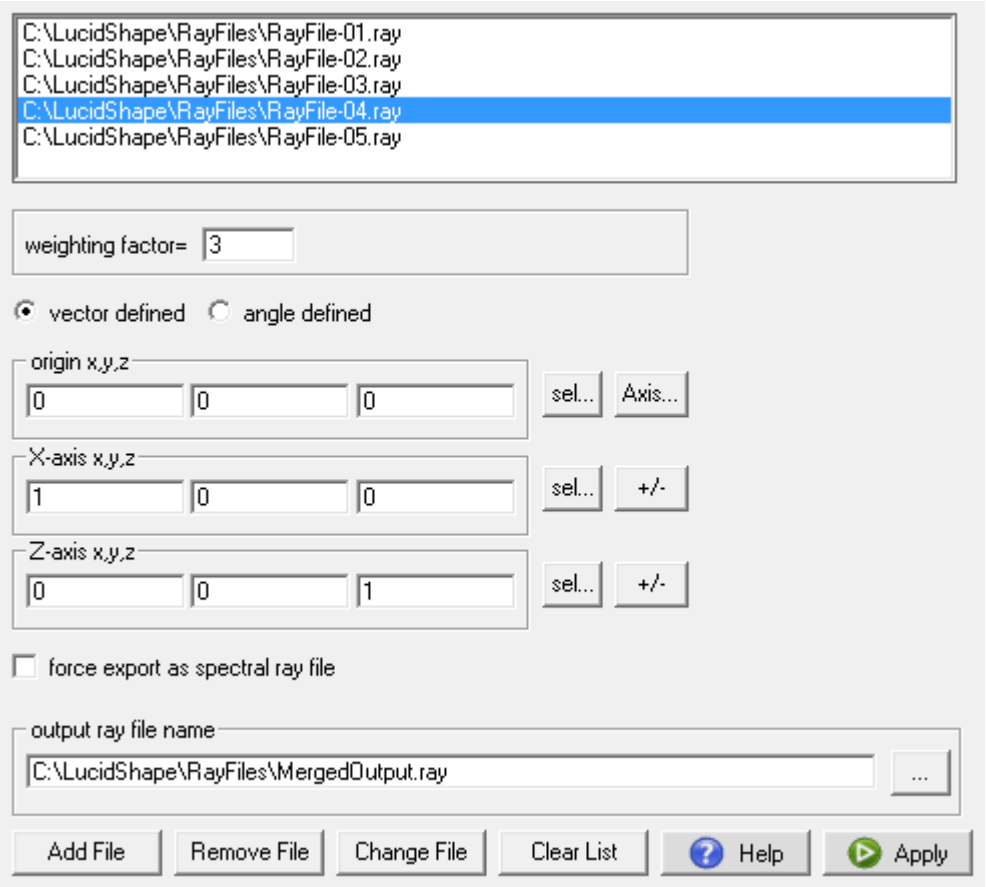


Figure 6.19: The Merge RayFiles Dialog

6.6.6. Compute a RayFile's Pseudo Focal Point (PFP)

This command computes a pseudo focal point (PFP) for a ray file and shows the information in the MessageView.

Given a set of rays and a point, one can compute the points' average squared distance from the rays set, that is the sum of all squared distances between the point and the rays. A ray set's PFP is a point with minimized average squared distance from the ray set.

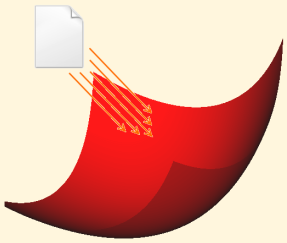
The resulting point and its average squared distance is displayed in the message view. Also, the average ray direction is computed and displayed.

input ray file name	The path and the name of the ray file of which the PFP shall be computed.

6.6.7. Project RayFile on Geometry

This tool takes all rays of an input ray file, computes their projected rays with respect to a given scene, and writes all projected rays to an output ray file.

If a given ray intersects with surfaces in a scene, the *projected* ray is defined as the ray with the intersection point as start point and the input ray's direction as the direction.



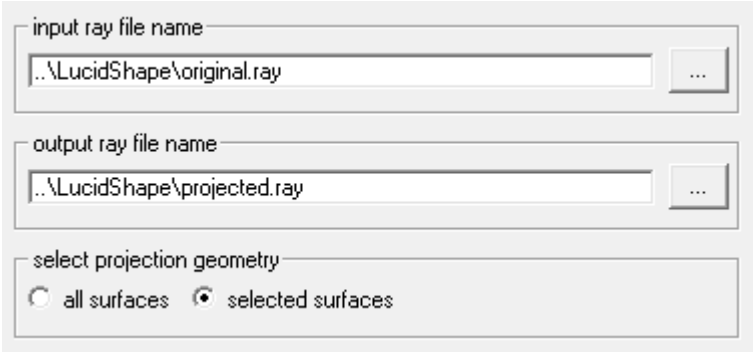


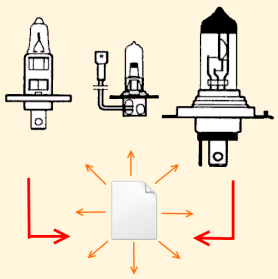
Figure 6.20: The Dialog for Project Ray File on Surface

input ray file name	The path and the name of the input ray file.
output ray file name	The path and the name of the output ray file.
radio button: select projection geometry	Selector which determines the surfaces in the model to take part in the projection computation.

6.6.8. Lamp Models for RayFiles

The lamp model library of LucidShape contains a set of predefined lamp models (..\LucidShape\lampModels\). Each lamp model consists of a set of emitting and absorbing shapes. One can either create ray files for automotive or general lighting.

You may view the lamp model library and create ray files from the models. When the ray file is created it may be tested, the ray file then can be used in future simulations.



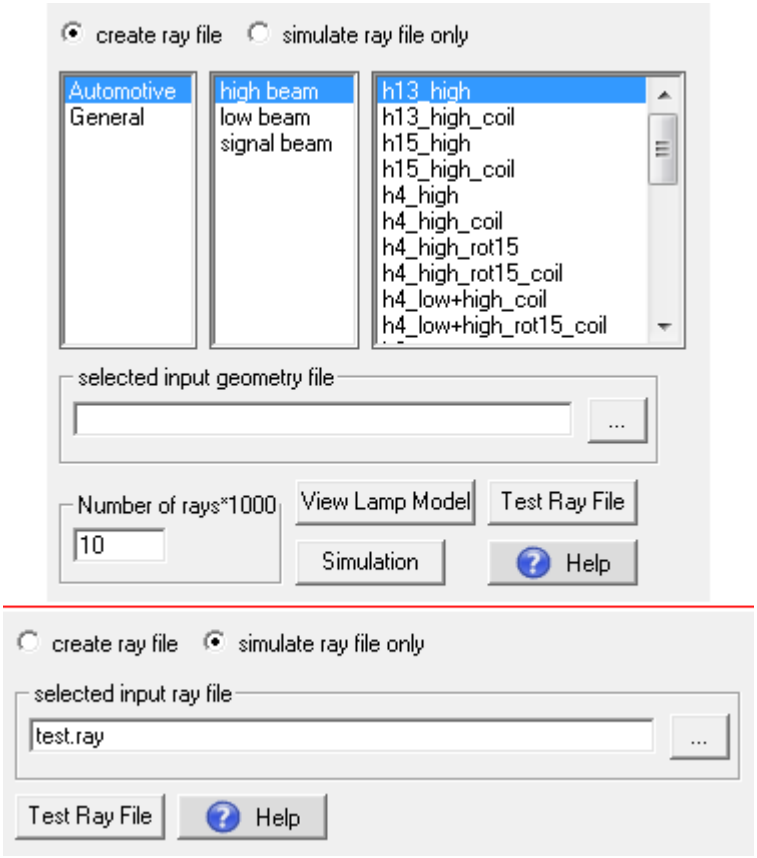


Figure 6.21: The Lamp Model for Ray Files Dialog

radio button: create ray file / simulate ray file only	Switches between the creation of a ray file or the simulation of a ray file.
listing	The choice is done via a hierarchical structure in the three selection lists, each list is sorted in alphabetical order. For automotive lighting there are the main groups: High beam, low beam and signal beam. For general lighting there are the main groups: Fluorescent, Metal Halide and Sodium.
selected geometry file	One can load any geometry file by using this dialog box.
*number of rays * 1000*	The number of rays to be used for the simulation.
button: view lamp model	Creates the selected lamp model.
button: simulation	Performs a simulation with the selected model and creates a ray file.
button: test ray file	Checks the ray file (a simulation has to be performed before that).
selected input ray file	The path and the name of the ray file.

Chapter 7. Emitter, Actor and Sensor Materials

Different material types can be created or assigned to the geometry in order to make a shape. Recall, a shape consists of a geometry and a material. The material properties and parameter are described each in the section Emitter Materials, Actor Materials, and Sensor Materials.

It is possible to assign more then one material to one geometry, but this always has to be handled with great care. If e.g. a shape has both a sensing and a reflecting material assigned to, there are two possibilities for a ray. Either it can intersect with the sensor first, because the sensor does not deflect the ray in any way it continues its way and intersects with the reflector. In the other case the ray may intersect with the reflector first, it is being deflected and thus nether intersects with the sensor. Because it is unpredictable which case occurs the consequence may be a non-uniform LID in the sensor even if used a complete homogeneous light.

In case it is really necessary to have a sensor directly on a reflecting shape, better use the possibility to create an offset geometry and assign the sensor to this surface. This additional surface can be seen as kind of a second layer of the reflector but the geometries not lying upon another and thus there is no problem with the order of the single intersections.

A physical and mathematical concept of surface properties (reflection, refraction, scattering, transmission, absorption) can be found in the Technical Reference Manual (..\LucidShape\doc\LucidTechnical.pdf).

7.1. The Dialogs *Create Material* and *Assign Material*

There is one dialog for each of the three groups of material: *Emitter, Actor* and *Sensor*. Each of these dialogs contains several predefined materials for the respective type of behaviour. When pressing *Apply*, the new material is added to the experiment (it is shown as a new entry in the TreeView, listed in the folder 'the whole file ...'), now it can be assigned to a geometry in order to create a new shape.

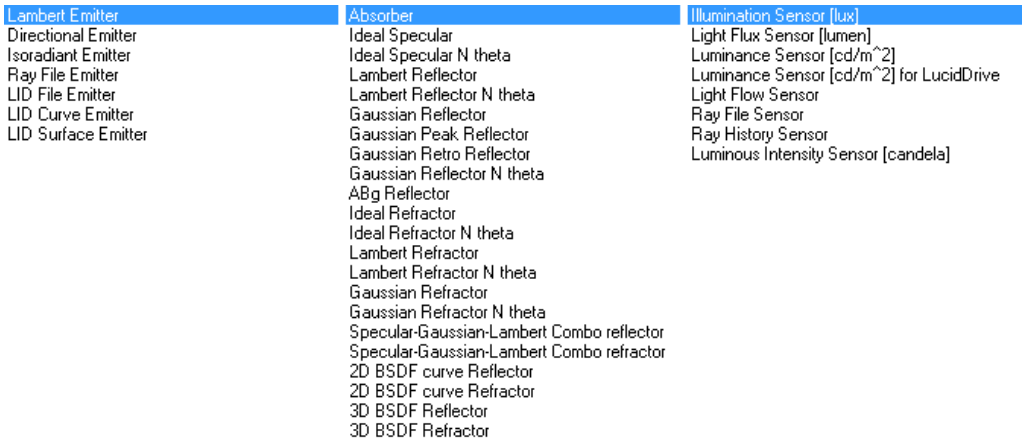


Figure 7.1: From left to right: Emitter-, Actor- and Sensor-Types

The *Assign Material* dialog is used for assigning a material (either an already existing one or one just to be created) to one or more geometries. There are several options to specify the geometries which shall be combined with the material to create a new shape (see the picture right below).

Figure 7.2: The part apply to of the Assign Material Dialog

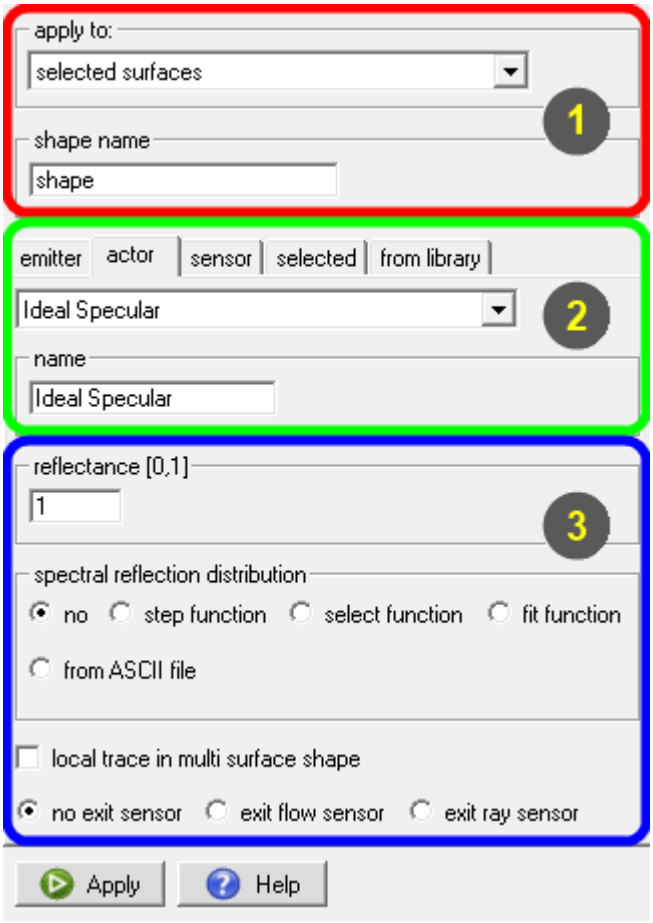


Figure 7.3: The Assign Material Dialog

1. *Geometry Part*: Here the geometries to be used can be chosen and the name of resulting shape can be set.

2. In this part of the dialog the general behaviour of the material can be chosen. You can select one of the three basic types: *Emitter* (to create light sources), *actor* (to create reflectors or lenses) and *sensor* (to create light detectors) and then choose from a list specific to that type.

   With the tab *selected* it is possible to assign a self-created material, with the last tab one can choose a material *from library*.

3. The third part is specific to the material type selection in part two. These dialog parts will be explained in detail in the subsequent sections.

The basic difference between the *Create Material* and the *Assign Material* dialog is that the first only creates a material and adds it directly to the model (for further use), while the latter creates a material and adds it to a user selected set of surfaces or shapes, to create new shapes which are in turn added to the model. *Assign Material* thus requires one or more existing geometries to assign to.

Additional there is a separate dialog *Create Medium*. This dialog can be used to create a medium and later on replace an already existing medium in a shape, it is not meant to create a material 'medium' which can directly be assigned to a geometry.

When the assigning process seems not to work: As usual, please have a look into the MessageView. Here LucidShape will show warnings or error when the current operation was not possible. May be the selected entry in the TreeView does not match the selected entry in the dialog.

7.2. Emitter Materials

Different emitter types can be assigned to the geometry in order to make a light source. Basically, an emitter is defined by the following options and parameters:

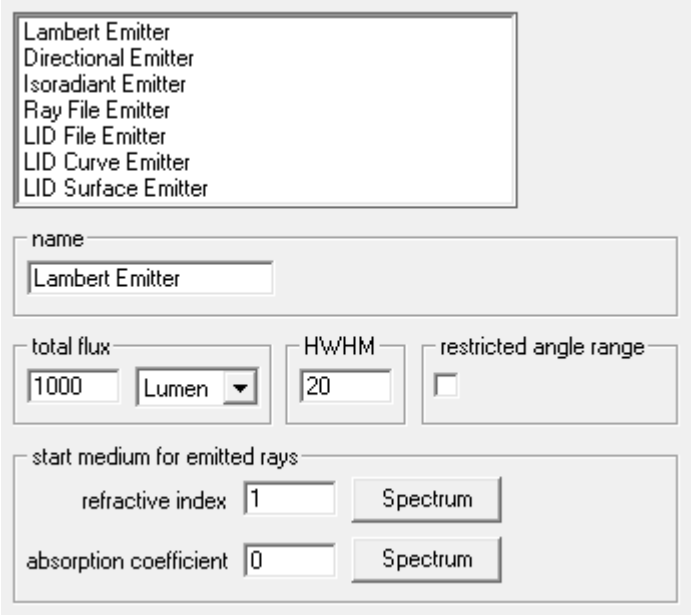


Figure 7.4: The Create Emitter Dialog

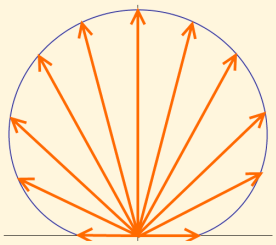
list	This list shows the basic emitter types, they are explained in detail in the following subsections.
total flux	Determines the total power output of this light-source, given either in [lm] or in [W].
half width at half max (HWHM)	Measured in the profile of the beam, the value determines the distance between the centre of the outgoing light and the point at which the power is down to 50% in the profile, this gives the spatial distribution of the outgoing light. (E.g. a very small value refers to a almost laser-like beam, whereas a very high value means almost parallel light.)

	Please refer also to Section 6.1, “Base Options and Parameters” for both a declaration and a sketch of HWHM.
flag: restricted angle range	If checked it is possible to constrict the outgoing light, this is independent to the profile of the beam (→ see HWHM right above). The positive value R cuts off the cosine emission outside the angular range [-R,R].
start medium for emitted rays	The refraction index attached to all emitting rays. Use the default value (1.0 for "air medium") for all setups without refractors. An index different from 1.0 indicates that the emitter resides in a medium with that index. Accordingly, the rays will be displayed as dotted lines inside the medium and straight lines once they pass any refractor surface. The absorption coefficient determines the absorption of the medium when starting in one.
spectral power distribution (SPD)	Here the spectral power distribution can be specified, explained in detail in Section 7.2.8, “Spectral Power Distribution (SPD)”.

The emitter types and their respective specific parameters are described in detail in the following subsections.

7.2.1. Lambertian Emitter

The light is emitted with a Lambertian distribution, i.e. a cosine distribution around the surface normal. A Lambertian emitter surface appears always with the same brightness to the eye of the beholder.



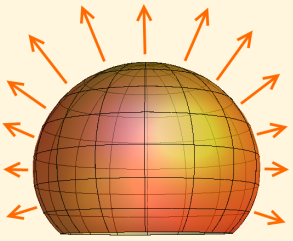
Beside the default setting, the beam spread (cosine distribution) can be reduced or enlarged by an exponent N. The distribution has the form: cos to the power of N. $I=I_0 \cdot \cos^N(\alpha)$. The beam half angle is defined as the angle with 50% intensity. It is calculated via

$\alpha_{1/2}=\arccos(0.5^{1/N})$

A HWHM value of 60 complies with the standard lambertian emitter.

7.2.2. Directional Emitter

This light emitter emits light always into the direction of the surface normal.

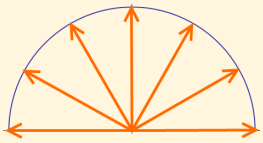


Apart from the parameters (flux, efficacy, start refraction index) described above, it has no additional parameters.

7.2.3. Isoradiant Emitter

This light emitter emits light in all directions with an equal intensity. It can only be used in combination with a point light source.

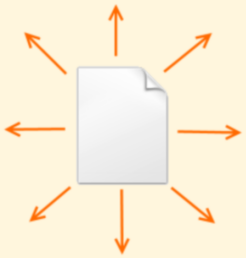
Apart from the parameters already described above (flux, efficacy, start refraction index), there are no additional parameters.



7.2.4. Ray File Emitter

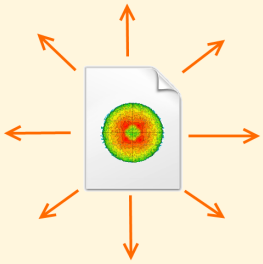
An emitter type uses a LID saved in a ray file. This ray file may be created by a previous simulation.

For the use of ray files please see also Section 6.6, “RayFile and RayFile Tools”.



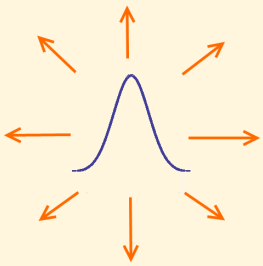
7.2.5. LID File Emitter

An emitter type which uses a LID saved in a LID file, which means that during the ray trace process, this emitter object gets the light intensity distribution from the given LID file. This LID file may be obtained either internal (from a previous simulation) or external (by importing a LID created by another software). This emitter can be used only in combination with a point light source.



7.2.6. LID Curve Emitter

Light emitter with a light intensity distribution taken from the z-component of a control curve to be chosen. This emitter can be used only in combination with a point light source.



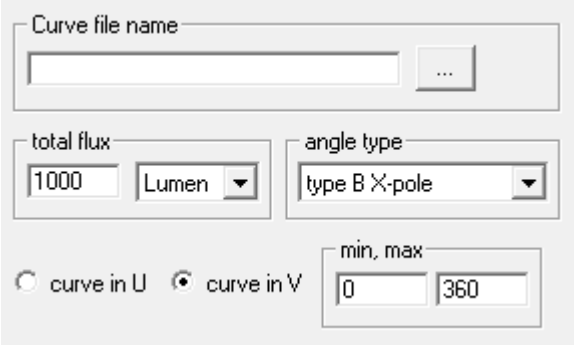
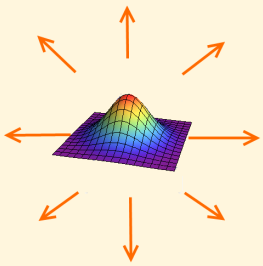


Figure 7.5: The LID Curve Emitter Dialog parts

LID curve file name	The path and the name of the file which holds the curve to be used.
angle type	Defines the angle type: B type (X-pole), A type (Y-pole) or C type (Z-pole).
radio button: curve in u, v	Determines whether the given curve is defined in u- or in v-direction.
min, max in orthogonal direction	Defines the uv-range depending on the usage of the curve: If the curve is used in u-direction it defines the v-range and vice versa.

7.2.7. LID Surface Emitter

Light emitter with the light intensity distribution taken from the z-component of a given surface. This emitter can be used only in combination with a point light source.



LID surface file name	The path and the name of the file which holds the surface to be used.
angle type	Defines the angle type: B type (X-pole), A type (Y-pole) or C type (Z-pole).

7.2.8. Spectral Power Distribution (SPD)

Beside the basic type of the emitter it furthermore can be specified via the SPD, this determines the spectral behaviour of the resulting light-source. The options and parameters are explained in the following table, for further and more detailed information (e.g. about the step or fit function and the correct use of the channels and the number of values which have to be entered) please also refer to the *LucidTechnical.pdf* mentioned at the top of this chapter.

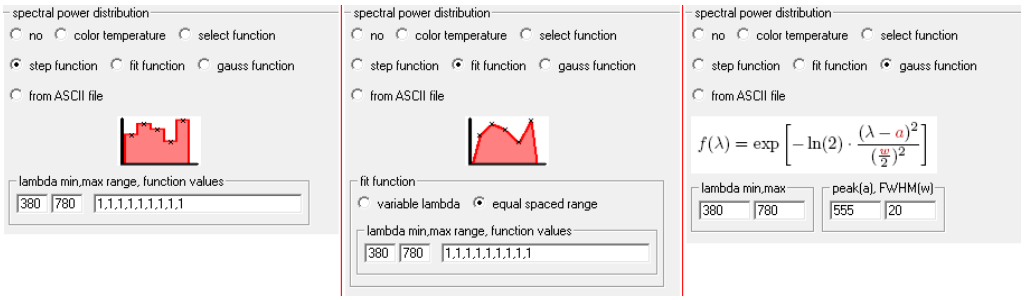


Figure 7.6: The Spectral Power Distribution (SPD) Dialog with different options

no SPD	No SPD is assigned to the emitter. The ray tracing will be performed as if there was monochromatic light - but without a specific wavelength. This means that rays are treated as purely geometric (i.e. classical ray optics). If a ray file emitter is used and the ray file contains SPD information, the SPD is taken from the ray file. This choice yields no further dialog elements.
color temperature	A black body color temperature is chosen for the spectral distribution function. The intensity distribution is a function of temperature, the higher the temperature, the shorter the wavelength of the intensity maxima. The color temperature is always written in units of Kelvin.
select function	A control curve can be selected from the geo view or tree view. See the dialog part "select function" in Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions” for a description.
step function	The spectral power distribution function is defined via a step function of equidistant steps. See the dialog part "step function" in Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions” or the *LucidTechnical.pdf* for a description.
fit function	The spectral power distribution function is defined as a piecewise linear function of equidistant steps (i.e. a linearly fitted step-function). See the dialog part "fit function" in Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions” for a description.
gauss function	A gaussian control curve can be determined by entering the respective parameters. After creating the emitter, the control curve parameters are still editable. λ_{min} *and* λ_{max} specify the wavelength range; *peak (a)* sets the location of the curve's maximum value, the location must lie in the wavelength range; *full width at half max (FWHM)* specifies the width of the control curve. *Degree, arcs, np* are used to specify the Gaussian curve's polynomial approximation: Degree = the polynomial degree, arcs = the number of polynomial arcs (a value of -1 means: this number is

	determined automatically), and np = the number of points used to approximate each arc.
from ASCII file	Instead of entering the values manually a complete file can be read. Each line in this file has to contain two values, the wavelength λ and its relative intensity in the range [0,1], separated by e.g. a space character or a semicolon. The whole file has to be saved as ASCII.

7.3. Actor Materials

Different "actor" material types can be created or assigned to the geometry in order to make a shape. Some properties repeatedly occur in reflective and refractive material types.

Many parameters are common to most or even all reflective and refractive materials, a fact which results in identical dialog elements. Instead of repeating the description of those elements every time, they are listed and explained in the following table, for the SPD please see the section right above.

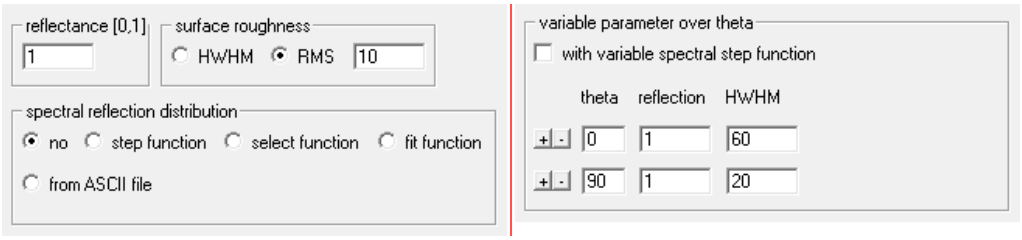


Figure 7.7: Some common options and parameters for a Reflective Actor

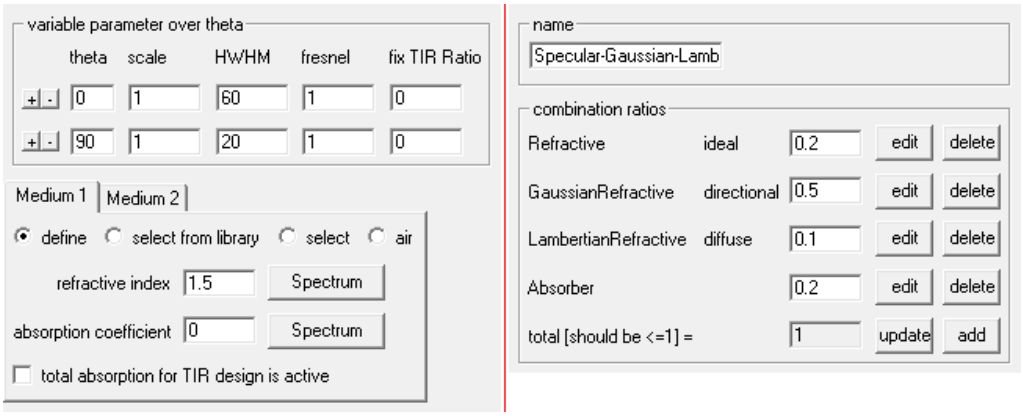


Figure 7.8: Some common options and parameters for a Refractive Actor

reflectance	Determines the amount of light to be reflected, the scaling parameter has to be in the interval [0,1]. A value of 0.0 refers to a perfect absorber, a value e.g. 0.8 means that every rays loses 20% of its current intensity. The reflectance can be specified either by one value for the entire wavelength range or spectral via the SPD settings.
spectral reflection distribution	This is nearly the same as the SPD, the only difference is that we now describe the spectral reflective behaviour. For further information please refer to Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions”.
theta	Theta is the incident angle of the incoming light, relative to the surface normal. To model a dependency on the incident angle it is possible to use more than one value for theta. (The number of lines can be changed via the '+/-' buttons, each line represents one term in a kind of fit function.)
flag: variable parameter over theta	If checked, the SPD can be specified for each θ too (additional to the already shown parameters).
surface roughness	Similar to the Lambertian distribution (see Section 7.3.3, “Lambertian (Reflective and Refractive)”) a parameter can be supplied that controls the width of the Gaussian curve (and effectively the "surface roughness"). Either a HWHM or RMS value may be set. The root mean square value σ is an angle, given in radian. In case of an elliptic behaviour, this value can set independently for u and v.
half width at half maximum (HWHM)	Measured in the profile of the beam, the value determines the distance between the centre of the outgoing light and the point at which the power is down to 50% in the profile, this gives the spatial distribution of the outgoing light. (E.g. a very small value refers to a almost laser-like beam, whereas a very high value means almost parallel light.) Please also refer to Section 6.1, “Base Options and Parameters” for both a declaration and a sketch of HWHM. Please note: The value HWHM only determines the strength of scattering of reflected (!) light (either reflected on a reflective surface directly or as Fresnel-reflection on a refractive surface). The transmitted light is scattered due to the settings for the HWHM too, but also influenced by the two refraction indices.
B,g parameters (reflective)	These two parameters determine the reflective behaviour for the ABg model.
scale (refractive)	Almost the same as the reflectance described just above but this time for the refractive actor, again the scaling value has to be in the interval [0,1].
fresnel (refractive)	This value determines whether a certain quantity of the incoming light shall be reflected due to the Fresnel equations, the value has to be in the interval [0,1] (with e.g. 0.0 means no fresnel reflection at all and 1.0 the fresnel reflection fully activated).

medium 1 and medium 2 (refractive)	An incoming and outgoing medium, defining an "interface" of refractive index contrast, e.g. a lens's behaviour, via Snell's law. Typical media are air (for the incoming medium) and glass (for the outgoing medium) to define the illuminated face of a glass lens. The outgoing face of the same lens would be defined by the interface from glass back to air. Any refractive "object" surrounded by air can be defined in LucidStudio this way by just two surfaces of refractive type - as long as they cover the complete ray paths in your system. This dialog part is described in detail in Section 7.5, “Media for Refractive Materials”.
flag: total absorption for TIR design is active	If checked the light usually reflected due to the TIR effect will be absorbed instead.
absorption coefficient (refractive)	The absorption coefficient α, this is not a percentaged value but determines the exponential decay of intensity in an absorbing medium due to the Beer–Lambert law.
gaussian type (reflective)	Using the 'GaussReflective' option will produce a usual gaussian distribution, while the use of the peak reflector results in a very sharp gaussian ('peak'). Both variants are also available as elliptic choices, for a more detailed description of the difference in the formulas used, please refer to the *LucidTechnical.pdf* chapter 2.4.
flag: local trace in multi surface shape(refractive)	When performing a Monte Carlo simulation using the *NURBS* method, the raytracer checks all surfaces whether they are hit by the next intersection. When a ray has entered a medium (i.e. it intersects with a MultiSurfaceShape) and the flag is checked, the raytracer only tests surfaces from this MultiSurfaceShape for the next intersection. So the flag can help saving some calculation time.
radio button: exit sensors (reflective + refractive)	The flag *exit flow sensor* adds an additional flow sensor material to the geometry; the flag *exit ray sensor* adds a ray file sensor to the model, both will collect all rays that hit the geometry.

Material Combination:

With this type of material the user is able to create an own material combination with the wanted properties in the wanted ratio (available for both reflective and refractive), the values can be set independently.

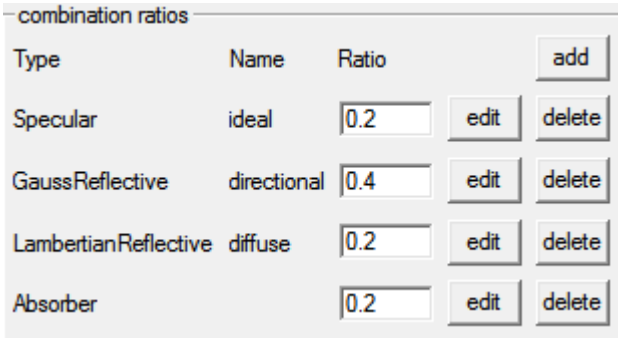


Figure 7.9: The Material Combination Dialog (both reflective and refractive)

button: edit	If pressed a small dialog for this material pops up, this is to edit the respective entry in the combination list.
button: delete	If pressed the respective line in the combination dialog is deleted.
button: update	If pressed the sum of the entered values is updated.
button: add	If pressed an additional entry may be chosen and added to the combination list. This way it is even possible to create a material with both reflective and refractive behaviour (e.g. a semi-transparent plane).

Volume Scattering:

In addition to the behaviour of single surfaces, a volume scattering can be specified. The user either can import a self-designed model or choose among three physical models and adjust the parameters to fit the customer-specific materials. Basically the *Henyey-Greenstein*-Model is the same as the *Isoradiant*-Model, only enhanced by an additional parameter.

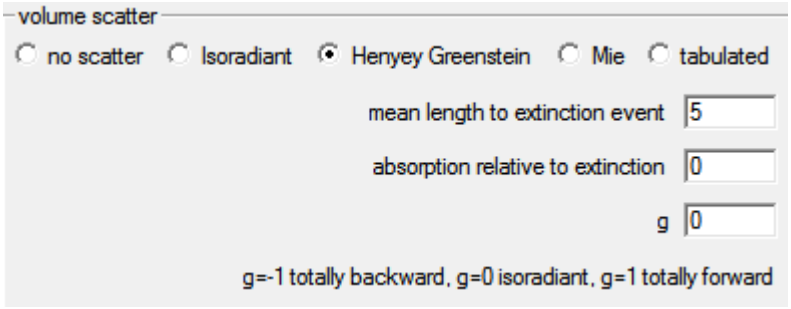


Figure 7.10: The Volume Scatter Dialog

no scatter	If selected, the volume scattering is deactivated (this is the default setting).
mean length to extinction event	Determines the mean free path of a ray between two intersections.
absorption relative to extinction	Quotient of absorption and extinction, this is the relative absorption to all extinction events (extinction is the sum of absorption and scattering). The value to be entered has to be in the interval [0,1], a value of 1 e.g. means that absorption is the only reason for loss of light in the system.
g	G determines the main direction and distribution of the scattering. A value of 0.3 for example indicates a mostly isoradiant distribution with the forward direction emphasised.
complex refractive index particle	The Mie scattering model uses a complex refractive index, to be entered as two values n_R, n_I.
particle radius	The radius of the scattering particle in [nm].
particle number	Determines the particle density as quantity per cubic millimetre.
lambda	When performing a spectral simulation, the function depending on the model is used; this parameter determines the value for λ to be used when performing a non-spectral simulation.

Technical amendment: Henyey Greenstein in LucidShape

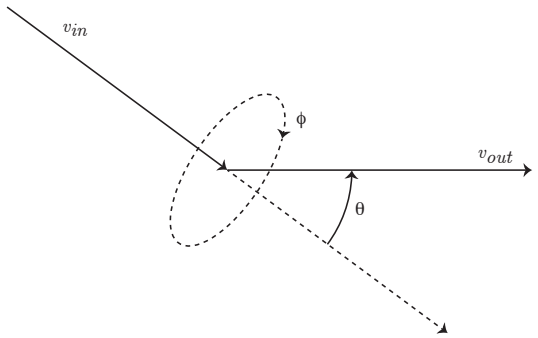


Figure 7.11: Henyey-Greenstein Scattering Model graphical

Using the notation of the graphic above, the Henyey-Greenstein model is implemented in LucidStudio as follows: v_{in} and v_{out} are the incoming and outgoing ray respectively; θ is the angle between v_{in} and v_{out} and ϕ is the rotation of v_{out} around v_{in}.

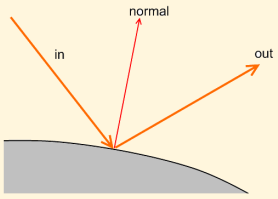
The Henyey-Greenstein volume scattering function is defined as

$$HG_{\cos\theta}(\cos\theta) = \frac{1-g^2}{2(1+g^2-2g\cos\theta)^{\frac{3}{2}}} \tag{7.1}$$

$HG_{\cos\theta}$ is a probability density, describing the $\cos\theta$-values of the scattered ray with respect to the incoming ray. After a ray is scattered towards a new direction determined by the $\cos\theta$-value, the scattered ray will be rotated around the incoming ray by ϕ, where the rotation by ϕ is equally distributed.

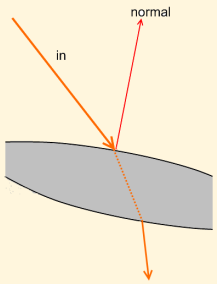
7.3.1. Ideal Mirror (Reflective)

A simple mirror, defined only by the (perfectly specular) reflectance described above.

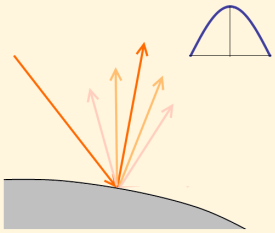


7.3.2. Ideal Lens (Refractive)

An ideal refractive material defines the specular refraction of light at the transition between two media of different refractive index.



7.3.3. Lambertian (Reflective and Refractive)



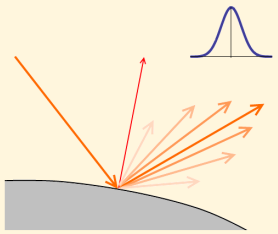
Lambertian reflective materials in LucidShape reflect the light diffusely rather than specular. A Lambertian surface reflects the light with a cosine distribution, where the cosine maximum is achieved along the direction of the surface normal, not along the direction of specular treatment of reflection. (This is different in the Gaussian Model where the outgoing ray obeying the law of reflection is marking the center of the scattered light distribution.) Usually, this type of behaviour is found only on strongly diffusive materials such as paper or rough surfaces with strong aspect ratios and wide scattering angles.

Lambertian refractive materials in LucidShape scatter the incoming ray after the refraction with a cosine distribution. The maximum of this distribution is aligned with the surface normal of the refractive surface, not with the expected direction of an ideally transmitted ray (no impact of Snell's law). (This is different in the Gaussian Model where the outgoing ray obeying Snell's law is marking the center of the scattered light distribution.) Usually, this type of behaviour is found only on strongly diffusive materials such as milky glass or materials with heavy volume scattering (e.g. using paints).

A Lambertian material is intended to be used only with high values of the HWHM angle (standard setting is 60°). It will show unexpected behaviour for small HWHM angles (e.g. 20°) combined with non-normal incidence. In combination with other materials it can complete a so-called combo material, adding a diffuse background with invariance to the angles of incidence to any optical response from a material (see Section 7.3.6, “Lambert/Gauss Combination (Reflective and Refractive)” and following).

7.3.4. Gaussian (Reflective and Refractive)

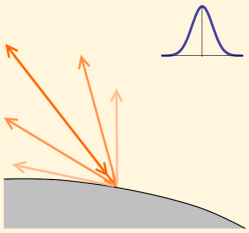
A surface with a Gaussian material reflects resp. refracts the light with a Gaussian intensity distribution. This works similar to a Lambertian material, but the distribution function is represented not by a cosine but a gaussian curve.



7.3.5. Gaussian Retro-reflex (Reflective)

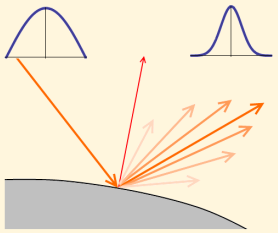
This material is similar to ordinary Gaussian reflective material, but the main outgoing ray's direction is always identical to the incoming ray's direction (instead of being determined by the classical law of reflection). The gaussian distribution function works with respect to this direction (where it achieves its maximum). This effect is realized by small refractive elements on the surface.

All parameters are the same as with the ordinary Gaussian reflective material, no refractive version is available here.



7.3.6. Lambert/Gauss Combination (Reflective and Refractive)

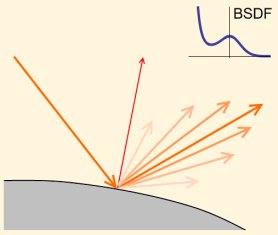
Combines Lambertian and Gaussian material to a combo material, with the combination ratio an individual material can be created (available for both reflective and refractive type of actor).



combination ratio	The ratio of the different types may be set independently to each other, the sum of all values should be equal to 1.0.

7.3.7. 2D BSDF Theta Curve (Reflective and Refractive)

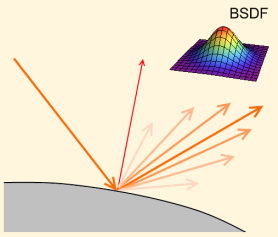
Builds a reflective or refractive actor material by direct application of a BSDF control curve, which may either be picked from the tree view or created via the BSDF wizard (see Section 7.7, “The BSDF Wizard”). The 2D curve is a function of zenith angle θ in spherical coordinates, and it is the same for all azimuth angles of incidence ϕ. Thus, it exhibits rotational symmetry around the axis of normal incidence.



BSDF curve definition	Determines how to select the BSDF curve. *select curve* reveals an object selector to pick an existing control curve from a tree view. The *BSDF wizard* is used to create a BSDF curve from material measurements, see Section 7.7, “The BSDF Wizard”.

7.3.8. 3D BSDF data from file (Reflective and Refractive)

Builds a reflective or refractive actor material by reading 3D BSDF measurement data (in LucidShape .lid format). These contain also lateral scattering information (azimuth angle).



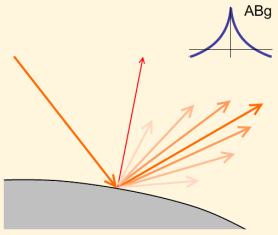
Selected input BSDF file	File input dialog for reading the measurement .lid data.

7.3.9. ABg (Reflective)

A reflective material according to the *ABg reflective model* which is an inverse power law model. The name is due to the three parameters in the BSDF equation, A,B and g, the coefficients of the BSDF-Model formula:

$$BSDF = A/(B + | \Delta \beta |^{g})$$

See the Technical Reference for more detailed informations about the mathematics.

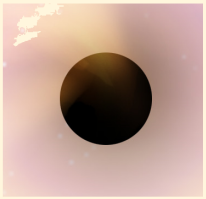


B, g parameter	The coefficients B and g of the BSDF-Model formula.

7.3.10. Absorber

An absorber material: A surface equipped with this material works as a "black hole" for all light rays.

The corresponding dialog has no parameters at all.



7.4. Sensors and Sensor Materials

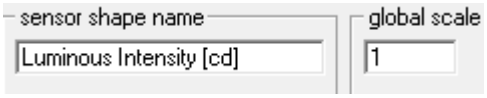
This section describes the various sensors types which are available in LucidShape.

Most sensors are from their very nature rectangular arrangements of light cells and may be attached to a geometry to count incoming rays. Special cases are the ray file sensor (which accounts rays into an external file to build ray files) and the volume sensor (a three-dimensional enhancement of the usual sensor concept).

There are also infinity sensors like the candela sensor which can not be applied to a finite surface, i.e. not to any of the shapes a user has created. These sensors are configured completely within their dialogs and use infinity surfaces (infinitely large, infinitely distant).

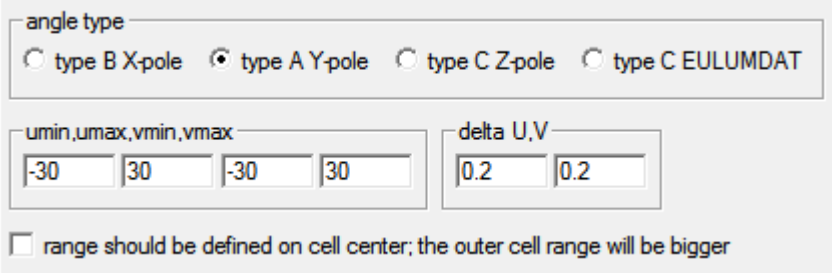
The parameters in the following are common to most or even all sensors:

- *Sensor shape name* and *global scale*: Here descriptive names may be entered, these names will be shown in the tree view too. A global scale factor gives the possibility to take global losses into account (e.g. due to fabrication tolerances). The default value is 1, which means no loss is assumed, a value of 0.7 for instance refers to a loss of intensity of 30% thus resulting in a sensing efficiency of 70%. (Please note: This linear sensor 'scale factor' may not be mismatched with the multiplicative 'efficacy scale factor' used for light-sources!)



- *Angle type*: Choose between three angle types: X-pole (type B), Y-pole (type A), Z-pole (type C) and Eulumdat (type C). For more information about the angle types, please see the LucidShape Technical Reference.

  The four values u_{min}, u_{max}, v_{min} and v_{max} determine the range of the sensor area, the two values $\Delta_{u/v}$ determine the resolution and thereby the subdivision i.e. the cell size of the sensor.

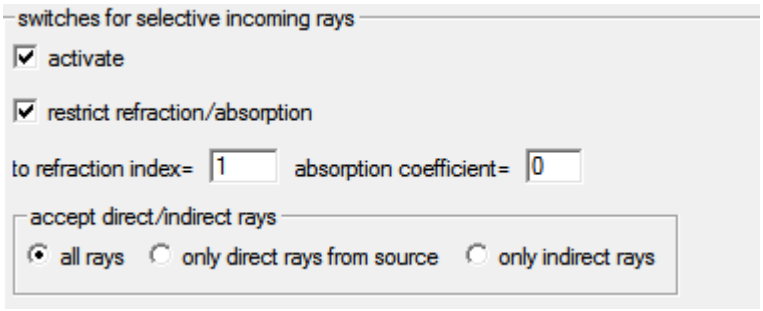


Range should be defined on cell center: Basically this flag changes the starting position of the complete sensor and thus the single sensor cells. Check this flag if you want to define the sensor range manually with the positions of the cell centers. For example: Usually, a full circle sensor is set up with the range of [0°,360°]. That is because LucidShape defines the sensor ranges by the edges which you want to have in your result and it compensates any overlap or exceeded cell sizes.

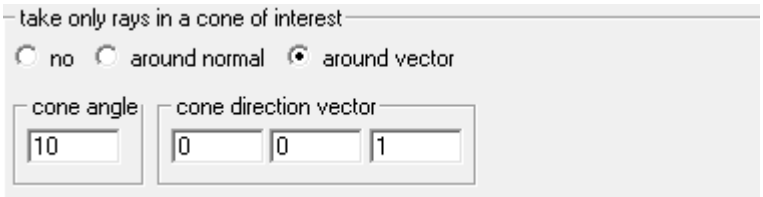
When using this flag, you have to specify the cell center minimum and maximum positions manually. With a cell size of $\Delta u = 1$, this would be [-0.5°,359,5°], building 360 cells, each 1° apart. This way, the first cell is located at 0°, the last one at 359°. Thus, no overlap is created.

Attention: A setting of [0°,360°] with $\Delta u = 1$ in contrast would result in 361 cells being created and 0° and 360° being the same cell (this will lead to double counts).

- *Switches for selective incoming rays*: If activated, the sensor filters the incoming rays and accepts only those rays for which the entered values for refraction index and absorption coefficient coincide. Also, flags can be set to only count direct rays (= rays which come directly from the light source WITHOUT ANY interaction) or indirect rays in the sensor (i.e. with at least one interaction).

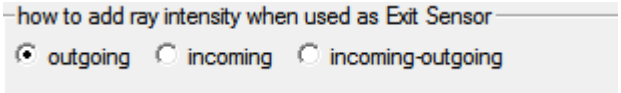


- *Take only rays in a cone of interest*: If enabled the sensor will count only those rays which intersect the sensor plane within an opening angle. The program calculates the angle between the sensor cell (more precise the normal at this intersection point) and the ray, if it is smaller or equal than the entered value (which is the radius of this cone) the ray is counted in the sensor, otherwise it will be ignored. One can choose either a cone which is orientated along the normal of this cell or along a specified direction vector.



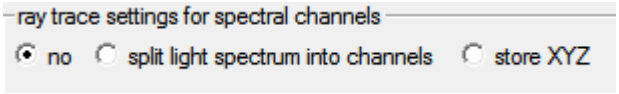
- *How to add ray intensity when used as Exit Sensor*: This radio button allows to toggle between three different modes. "Incoming" simple detects all incoming light which is intersecting the surface the exit sensor is assigned to, "outgoing" stands for the light which is leaving this surface (either refracted or reflected).

  In case the surface is not ideal (e.g. a reflector with a reflectance smaller than 1 or a medium with an absorption coefficient larger than 0), there is a difference between the incoming and the outgoing amount of light, this can be measured using the third mode "incoming-outgoing".



- *Settings for spectral ray trace*: The spectral settings can be deactivated if only performing non-spectral simulations). If set to *split light spectrum into channels*, the sensor will be split into a number of wavelength channels and each incoming ray will account for the channel that best matches the ray's wavelength. After a simulation, one can navigate through this 3D UVW-data by clicking the right mouse button and select the "Navigate 3D UV data" in the UV data view or using one of the spectral operations available under Menu → UVView → Spectral Calculations.

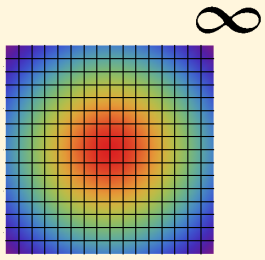
  When using the option *store XYZ*, the light automatically is converted, thus the LIDView is shown in an RGB format.



Please note: If any changes in the dialog settings have been applied, a new simulation has to be done.

7.4.1. Candela Sensor

The sensor detects the luminous intensity distribution [Candela]. An optical scale factor can be used for an overall scaling of the experimental setup. The result can be stored in any light data object like LIDData or IESData.



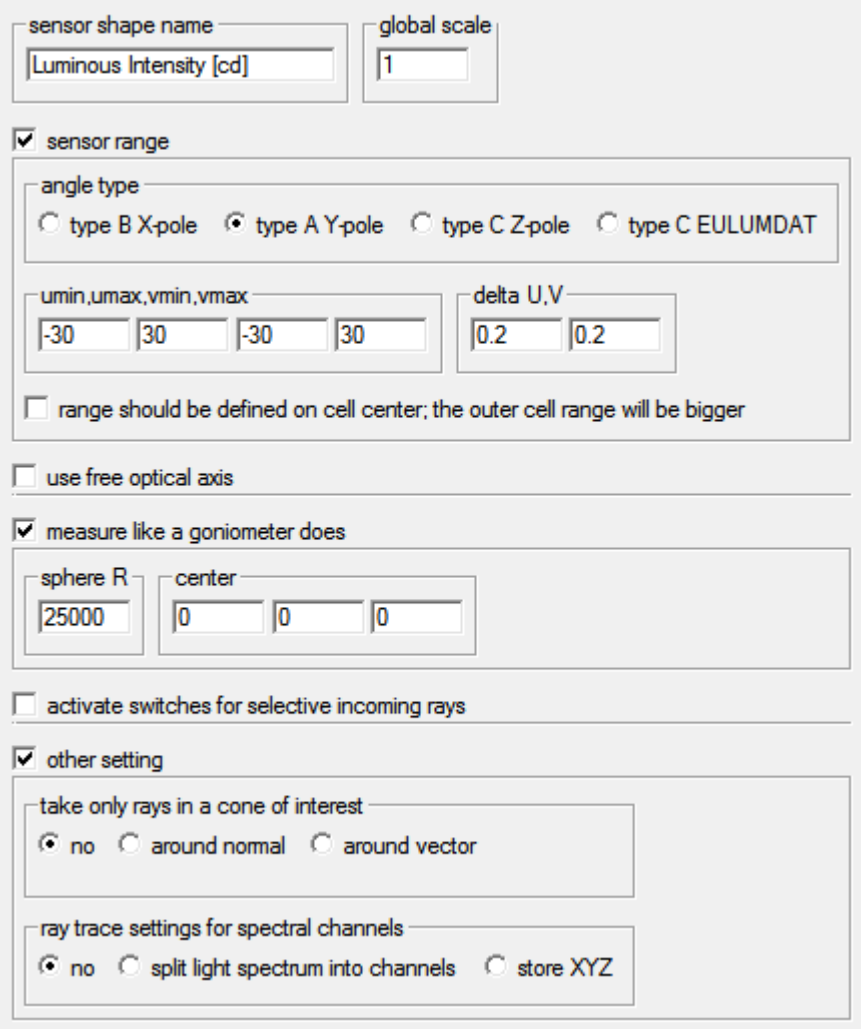
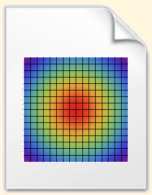


Figure 7.12: The Candela Sensor Dialog

global scale	A global scale factor in the interval [0,1] to take a global loss into account.
angle size [°]	The size and resolution of the Candela sensor in [°]. • u_{min}, u_{max}, v_{min}, v_{max}: The UV-range in [°]. • *delta u, v:* The cell size in [°].
use free optical axis	If checked, direction and base x,y,z-parameters can be entered. Internally the Infinity-surface is rotated thus defining a viewing direction.
measure like a goniometer does	If checked, measures like a goniometer, i.e. creates a sphere with radius R centered around a point (x,y,z) which is used as a detector for the incoming rays. The radius R and the (x,y,z) center may be supplied by the user.
settings for spectral ray trace	See Illumination and light flux sensor.

7.4.2. Ray File Sensor

This sensor captures all rays which are going through the infinity surface, i.e. they have left the complete geometry behind. The rays are stored either in binary- or in ASCII-format.



ray file name	The name and the path of the ray file, e.g. "sensor.ray".
storage type	Switch between ASCII text file or Binary file (LucidShape format).

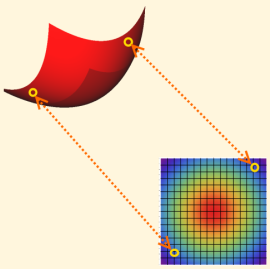
This sensor type can be used to record an own ray file from a given geometry. Recording the rays gives the possibility to kind of freeze the random generator (used in every usual simulation). That way the behaviour of the light-source is fully reproducible. In addition the characteristics of a light-source can be passed without revealing the exact structure of the geometry itself.

The physical unit stored in the resulting RayFile (either [lm] or [W]) is determined by the flag *simulate radiometric quantities* in the simulation dialog. Please also refer to Section 8.1, “Monte Carlo Ray Trace”.

The use of a RayFile (i.e. after being assigned to an emitter) is explained in Section 6.6, “RayFile and RayFile Tools”.

7.4.3. Ray History Sensor

The ray history sensor can be used to display rays in the geometry view which come from a particular region in the light intensity distribution. This analysis tool enables the user to backtrack glare and other problems in the beam patterns.



After creating the sensor, one should perform a Monte-Carlo ray trace. To display the rays in the GeoView, use the dialog "Restore Rays from Ray History Sensor", please also refer to Section 9.4.3, “Restore Rays from Ray History Sensor”.

Please see also the LucidShape demo "Ray History Sensor" at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/demos/.

ray history file name	The name and path of the ray history file, e.g. "RayHistSen.hst".
angle size [°]	The size and resolution of the Candela sensor in [°]. • u_{min}, u_{max}, v_{min}, v_{max}: The UV-range in degree. • *delta u, v:* The cell size in degree.

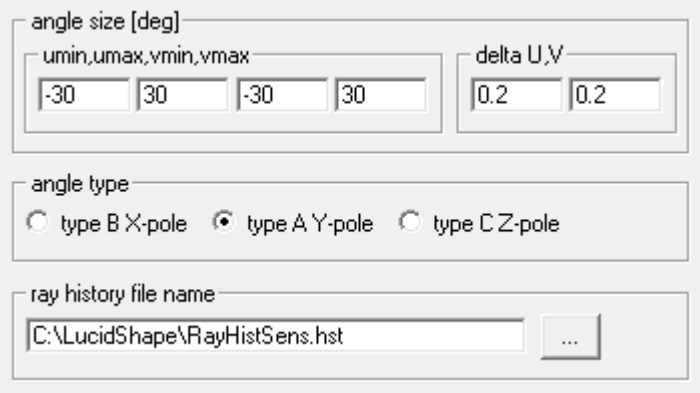
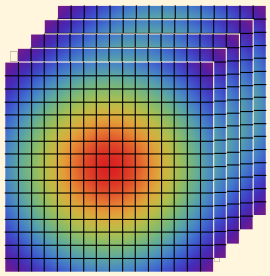


Figure 7.13: The Ray History Sensor Dialog

7.4.4. Volume Sensor

The volume sensor accounts rays in a three-dimensional box. This box is divided into single three-dimensional cells, so-called voxels. Various accounting methods are available.

After creating the sensor, one should perform a Monte-Carlo ray trace. To display the simulation result, the dialog "Visualize Volume Sensor" has to be used, see Section 9.4.2, “Visualize Volume Sensor”.



pull-down menu: sensor mode	Determines the ray detection mode. The *flux* mode counts all the light crossing the respective voxels, this is the default setting. The three modes *flux ortho x/y/z* are working like a lux-sensor, i.e. they are counting the light crossing the respective plane of the single voxels. The mode *absorbed flux* visualizes the absorbed light, e.g. in media with an absorption coefficient ›o.
size	Specifies the size and resolution of the volume sensor, x_{min}, x_{max}, y_{min}, y_{max}, z_{min} and z_{max} are the coordinates of the sensor box. *Delta x,y,z* determine the resolution of one cell in the respective directions. *Unit factor* supplies the basic units of the box (o.oo1 stands for millimeters).
fit model	Creates a box with a fitting range for the complete model.
fit object	Creates a box with a fitting range for the selected object.
extend	Like *Fit Model/Object*, but will not shrink the current volume sensor's box.
switches for selective incoming rays	Using the mode *absorbed flux* the refraction index has to be set according to the medium where the absorption takes place. (If not specified the volume sensor only shows data for the default value of the refraction index which is 1.o)

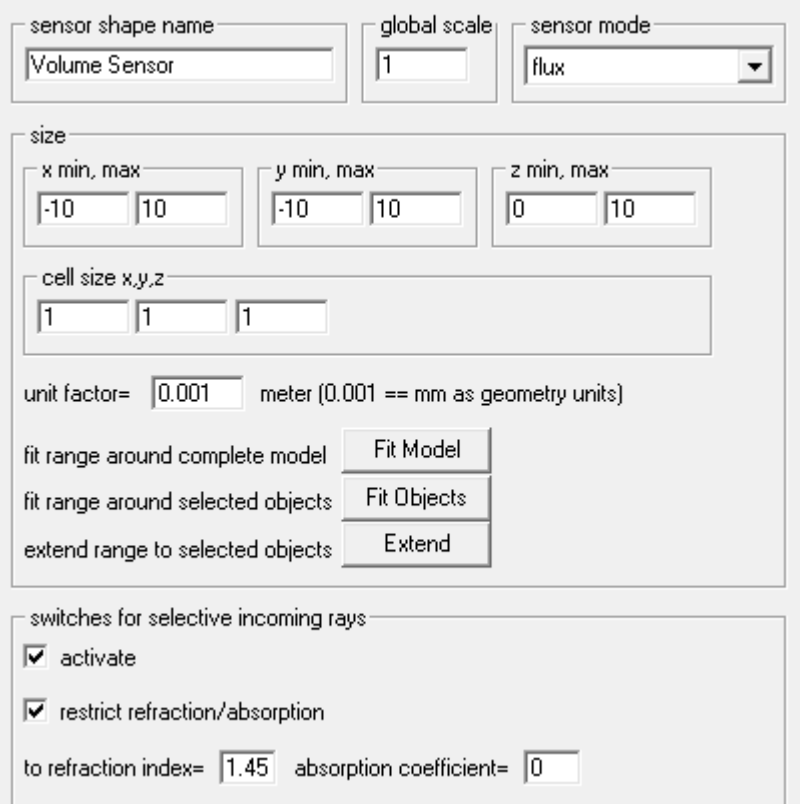
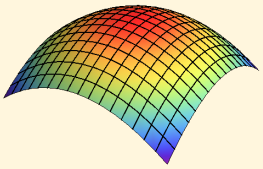


Figure 7.14: The Volume Sensor Dialog

7.4.5. Light Flow Sensor

The light flow sensor collects the light flow going through the base surface. The flow is stored in several angular light flux distributions per sensor cell.



The sensor surface is subdivided into n_u times n_v cells. You can change the effective sensor resolution indirectly via the chosen cell size (scaled by the unit factor) and the actual size of the geometry the sensor material will be applied to. This sensor setup is similar to the one of Section 7.4.7, “Illumination (Lux), Light Flux (Lumen) and Luminance (Candela per m2) Sensor Materials”.

For each cell, a light flux distribution is set up, storing the flux from/to different angles. The coordinate system for this is chosen from the A-, B-, C-type or from the local axes. The angular range of the flow sensor is set for all cells equally in a separately addressed u-,v-coordinate system from u_{min} .. u_{max} in steps of Δu (same for v, respectively). An optional scale factor can be used for an overall scaling similar to a sensitivity factor.

cell size	The cell size (its area) is the "inverse resolution" of the sensor, scaled by the unit factor. A cell size of "1" aims at a cell area of (unit factor)^{2}m^2 in the final shape. Lucid Studio will realize discretization of the final sensor as close as possible to the desired cell size.
global scale	A global scale factor in the interval [0,1] to take a global loss into account.
unit factor	A meter factor which scales the model units, e.g. the value of 0.001 scales to [mm] (millimeters).
dimension of light matrix of each cell (units in degree)	• *range in u, range in v:* The min/max angles of the allowed range of the flow sensor light distribution on each cell. The angular data is stored in a separate light matrix. • *delta in u,v:* The size of each angular element of the light distribution, affecting the angular resolution - angular 1D equivalents of the cell size.

The flow sensor may be used later to calculate the luminance [cd/m^2] at different viewing angles. The spherical coordinate system for the flow sensor is typically selected as an A system (see the Technical Reference), since the optical axis is always the z-axis, by LucidShape standard. In certain cases, like the flow through the curved outer parts of a lens surface, it is necessary to use a local axis system. Otherwise all the light that propagates at angles differing by more than the selected min/max values from the optical axis will not be measured. It is possible to have an asymmetric opening angle, e.g. (-50,-40) creates a 'cone of interest' at 45° sidewards with an opening angle of 5°.

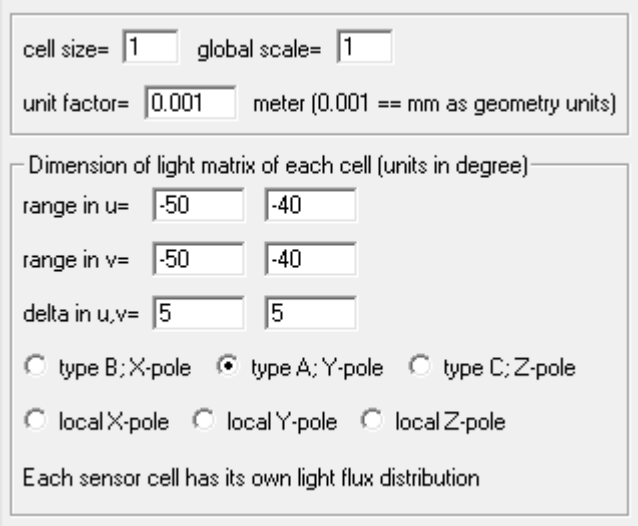
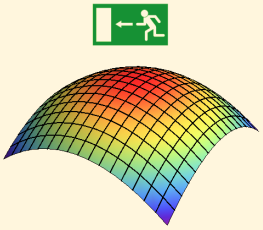

cell size= 1
global scale= 1
unit factor= 0.001
meter (0.001 == mm as geometry units)
Dimension of light matrix of each cell (units in degree)
range in u= -50 -40
range in v= -50 -40
delta in u,v= 5 5
type B; X-pole
type A; Y-pole
type C; Z-pole
local X-pole
local Y-pole
local Z-pole
Each sensor cell has its own light flux distribution

Figure 7.15: The Light Flow Sensor Dialog

7.4.6. Exit Sensors

Exit sensors are sensors which may be attached to shapes in order to collect light that exits the shapes during a Monte Carlo ray trace simulation.

Three types of exit sensors are available: flow sensors, ray file sensors (both available via a common dialog) and volume sensors (available via a dialog of its own).



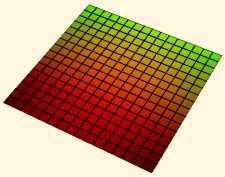
exit sensor type	Choose the sensor type to be attached to all currently selected shapes. Possible types are *flow sensor* (see Section 7.4.5, “Light Flow Sensor”) or *ray file sensor* (see Section 7.4.2, “Ray File Sensor”).

assign to	Here, the target shape set may be specified. Available choices are *all surface shapes*, *selected surface shapes* or *selected object vectors / sets*. For a description of volume sensors, see Section 7.4.4, “Volume Sensor”.
volume sensor	An object selection dialog to pick the volume sensor which is assigned to the selected shapes.

7.4.7. Illumination (Lux), Light Flux (Lumen) and Luminance (Candela per m^2) Sensor Materials

A sensor material which may be connected to a surface (usually a plane) to detect illumination in lux, light flux in lumen or luminance in candela per m^2. An optical scale factor can be used for an overall scaling of the experimental setup.

The illumination sensor also can be preloaded with a constant lux value. This may be used in reverse sensor light in order to calculate a LID for a point light source which results in exactly the same constant light distribution on the sensor.



[Lx] [Lm] [Cd/m^2]

cell size	The cell size of the sensor, scaled by the unit factor. A cell size of '1' aims at a cell area of (unit factor)^2m^2 in the final shape. LucidShape will realize this discretization of the final sensor shape's surface in the global coordinate system as good as possible.
global scale	A global scale factor in the interval [0,1] to take a global loss into account.
const [lx] / [lm] / [cd/m^2]	The sensor can be preloaded with a constant value, e.g. to simulate some stray light.

unit factor	A meter factor which scales the model units, e.g. the value of 0.001 scales to millimeters [mm].
pull-down menu: illumination type	Sometimes it is useful to collect "radial" lux. If this flag is set, the correction factor "cos(α)" is ignored in this case. This feature can be used e.g. for the bird's eye view of automobile head lamps.
reflection index (only for diffuse surface)	The factor 1/π, i.e. the ratio between luminance L [cd/m^2] and luminous intensity E [cd] for an ideal diffuse reflector. One does not have to divide by π again, the division is implemented in the calculation.
flag: emissive surface (only for LucidDrive)	If checked, an additional entry will be added to the respective surface or shape. When now the model is opened in LucidDrive, the this way modified surfaces are glowing. This is an optical effect only, no light is projected onto the road. (This is intended to show (i.e. slightly illuminate) the rear lights.)

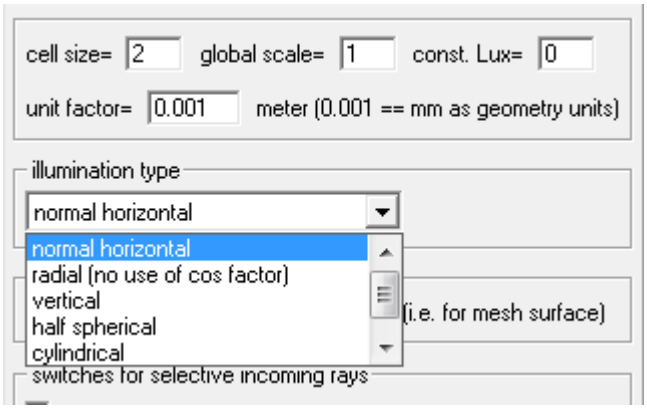
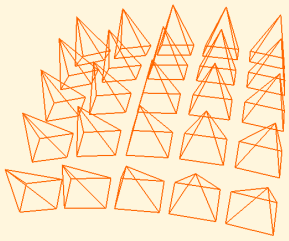


Figure 7.16: The Illumination, Light Flux and Luminance Sensor Dialogs

Please note: The sensor *sensor for diffuse surface* does have the same physical unit as a luminance sensor, but it is not a 'real' luminance sensor. This sensor material among others is intended for license plates. When the rays are intersecting with the sensor shape, this material behaves like a perfect diffuse surface. These scattered rays now are counted into the sensor.

7.4.8. Luminance Camera



The Luminance Camera is a special sensor type. You can simulate the appearance of your design to the human eye: The Luminance Camera Sensor captures the luminance images in your experiment like a real luminance camera would do.

In contrast to other sensors, you have to define a viewing direction and a location to look at. Also, it is possible to define more than one position and make the camera shoot images from several different angles (multi positioning).

The image field of this camera sensor is the virtual analogy to the CCD or photo-diode array of a digital camera. Width and height are its physical dimensions.

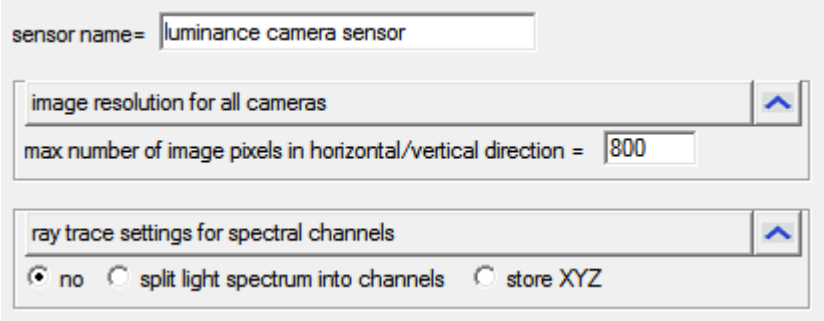


Figure 7.17: The Luminance Camera Dialog (I)

sensor name	The name of the luminance camera.
image resolution for all camera	Defines the image resolution in horizontal/vertical direction. The aspect ratio of the sensor image is taken from the field of views or the GeoView.
radio button: settings for spectral channels	This effect of this radio button is identical to the one in the usual candela sensor. The sensor can ignore spectral informations ('no'), count the light into the single channels or directly convert the recorded data into the XYZ format.

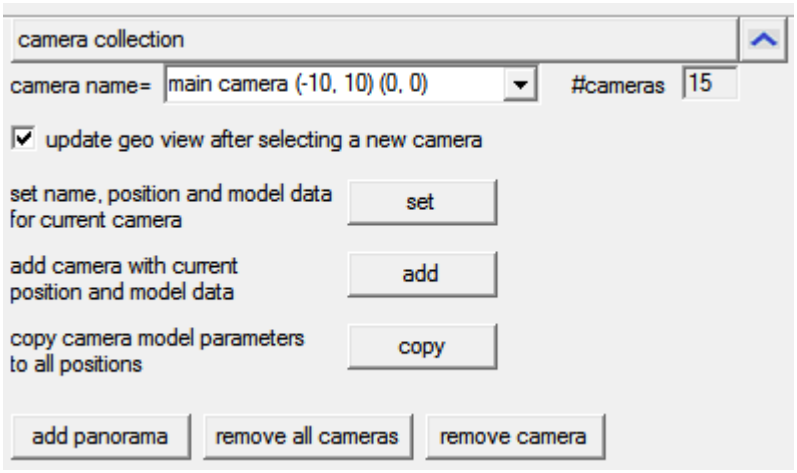


Figure 7.18: The Luminance Camera Dialog (II)

camera name	After selecting an entry via the drop-down menu, the respective camera can be renamed.
flag: update geo view after selecting a new camera	Check/uncheck this flag to update/not update the GeoView.
button: set	Applies the name, the position and the model data onto the model.
button: add	Adds a new camera with the current settings and parameters to the model.
button: copy	The camera parameters for a panorama have to be entered only once. Pressing this button will copy all parameters of the current camera to all positions.
button: add panorama	Creates a list of camera positions which form an angular scan, a panorama. This allows the rapid preparation of multi-position luminance images.
button: remove (all) camera(s)	Removes a single or all cameras from the list.

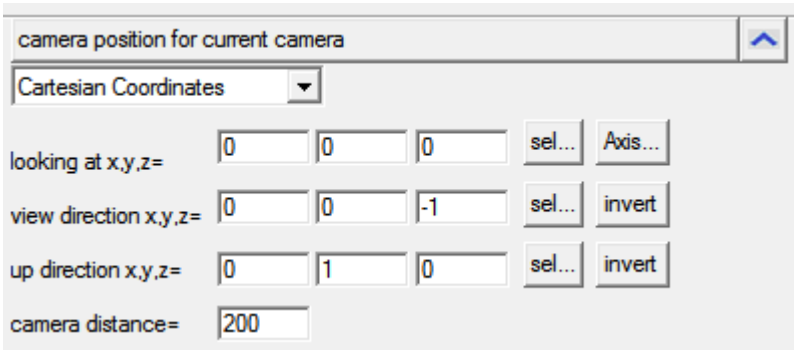


Figure 7.19: The Luminance Camera Dialog (III)

coordinate system	Defines whether a cartesian (x,y,z) or a y-pole coordinate system (gamma, tau) is used.
looking at	Defines the position (x,y,z) in the experiment to aim the camera at.
view direction	Defines from which direction the 'look at' position will be observed.
up direction	Defines the vertical axis to complete the orientation of the camera's coordinate system.
camera distance	Defines the distance between the camera (more precisely its lens or aperture) and the defined 'look at' position.

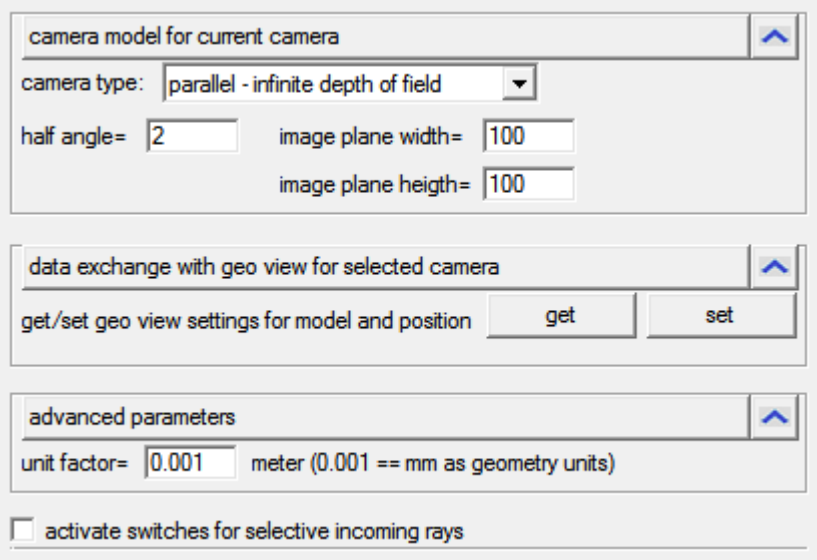


Figure 7.20: The Luminance Camera Dialog (IV)

camera type	Defines the type of this camera to either pin hole, thin lens or parallel.
half angle	This is the half of the cross-section of the solid angle, given in degree.
set stop	The aperture stop of the camera has to be defined: The f-number is the ratio of the aperture area and the focal length. It is also possible to select the actual radius of the stop. (In case of the parallel camera model, you can also define the cone of acceptance via the half angle directly.)
focal distance	Defines the focal length of the camera. When using the 'thin lens model' it is directly the focal length of the lens. In case of using 'pinhole', the focal length is the distance between the aperture and the image plane. When using the half angle in the parallel model, the focal length is not defined, the image plane then is identical to the camera plane.
horizontal, vertical FOV angle (field of view)	Defines the horizontal and vertical field of view respectively, both given in degrees.

image plane width, height	Defines the horizontal width and the vertical height of the image field respectively.
button: get/set	Exchanges the data with GeoView. Using the GeoView as a preview for the camera image - and then simply adopt the values from it into the dialog - is a rather easy way to adjust the parameters.
unit factor	A meter factor which scales the model units, e.g. the value of 0.001 scales to millimeters [mm].

For a more detailed treatment of the background of the Luminance Camera, especially the different camera types, please consult the Technical Manual.

7.4.9. License Plate Sensor

The License Plate Sensor is a sensor designed for the special requirements of a license plate. This sensor offers the possibility to check the illumination demanded for several specific regulations.

Regarding the license plate sensor, there are some subtleties worth mentioning:

- The unit the sensor shape does measure is cd/m^2. Although this is the same unit as for a luminance sensor, its behaviour is complete different. When the rays are intersecting with the shape of the license plate sensor, it behaves like a perfect diffuse surface (lambertian reflector, this is to emulate the particular surface of a license plate). These scattered rays now are counted into the sensor.
- Creating a license plate sensor leads to the creation of two shapes. One shape is the 'usual sensor', it is used just for showing the LID on the shape but not for the rating. The other shape is the dot sensor, this one is used for the rating with the test table.
- This dot sensor has as many cells as dots are shown in the GeoView (this number should match the respective regulation). This also leads to the rather gross resolution in the UVView.
- The sensor cells in LucidShape are rectangular, the dots in the regulation are circular, internal the size of the sensor cells is adjusted so they have the same size as the dots.

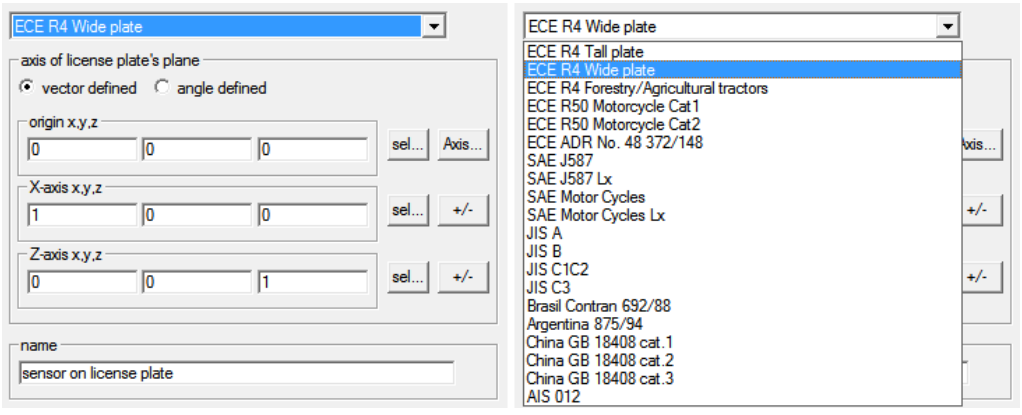


Figure 7.21: The License Plate Sensor Dialog

license plate type	This offers one to choose from a variety of different legal regulations.
vector/angle defined	This flag determines the type of the axis system.
origin x, y, z	Defines the coordinates of the origin.
x-axis/z-axis	Defines the coordinates of the x- and the z-axis respectively.
rotation angles about Z, Y', X''	Defines the rotation about the axis in this given order (after rotation about the z-axis the y-axis is transformed y → y', ...).

This paragraph shortly describes the setup of a license plate sensor onto an imported surface achieved from a CAD program. After importing the geometry, there are (among other) a sensor area (shown as blue in the picture just below), the license plate (red, in two parts with a mounting hole in each segment) and two cylindrical light sources (yellow).

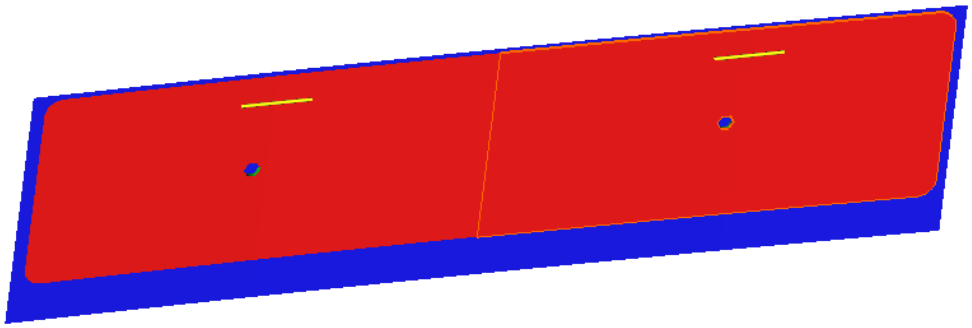


Figure 7.22: Simplified Geometry: The license plate area (red) in two parts (right one marked in orange), the sensor area (blue) and two cylindrical light sources (yellow)

Open the dialog via "Menu → Sensor → License Plate Sensor" and select the right test table. Click onto "axis" and double-click onto the license plate surface. If there is an axis system attached to this surface, it is taken to the "select axis". Maybe the orientation of the plate sensor has to be adjusted, depending on the orientation of the axis system received from the CAD program. The extra dialog allows to determine the position of the axis system on the surface, e.g. if it is located centre justified, in the lower left corner, etc.

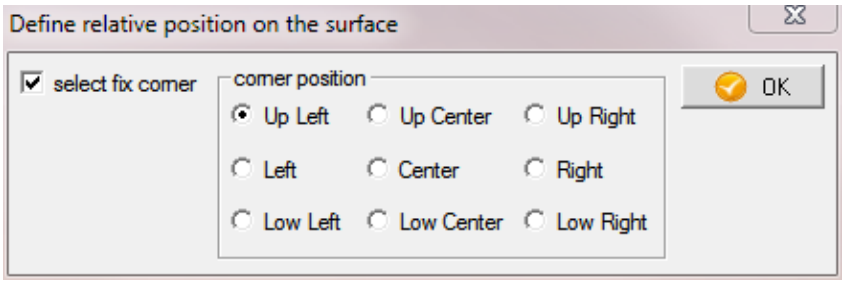


Figure 7.23: The same geometry with the two sensors created, the dots marked in orange

If the respective surface does not contain any axis system, it easily can be created via "Menu → Geometry → Create Axis System". Select "Axis ...", mark the surface and apply that, now a new axis system is attached to this surface (possibly the orientation of it has to be adjusted).

After applying the license plate dialog, two sensors are created. One plane senor to show the layer where the measurement will be done, and one sensor with dots, these dots indicate the single measurement points (marked in orange in the picture right below).

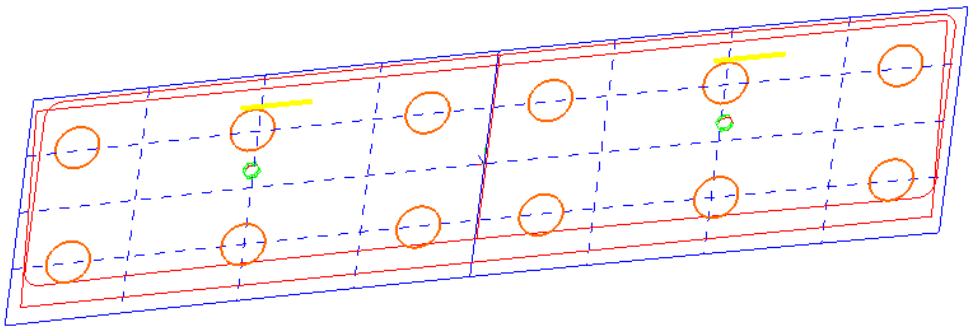
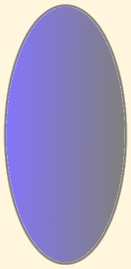


Figure 7.24: The same geometry with the two sensors created, the dots marked in orange

7.5. Media for Refractive Materials

Creates a medium to be used for refractive materials. The medium will appear as an object in the tree view, where it can be selected and assigned to a chosen material. For spectral simulations, detailed spectral distributions can be used, otherwise LucidStudio allows you to apply the average values common in literature, e.g. of the refractive index of a material in the visible spectral range.



There are two ways to add a medium into the model, either by selecting a medium from the media library provided in LucidShape or to define an own medium. The library is sorted in the categories glass (among other Hikari, Hoya, Schott, etc.) and plastic (among other Evonik). It is accessible via *select* → *select from library*.

The second way is to use the option *define* → *spectrum* to define the behaviour of the new medium.

medium name	The name of the created medium.
refraction index	The refraction index of the medium. This single value will be ignored, if a) a spectral simulation is performed and b) a spectral refraction distribution is set.
absorption coefficient	The absorption coefficient α, this is not a percentaged value but determines the exponential decay of intensity in an absorbing medium. This single value will be ignored, if a) a spectral simulation is performed and b) a spectral absorption distribution is set.
spectral refraction distribution	This is used in spectral simulations instead of the average index of refraction. See Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions”.

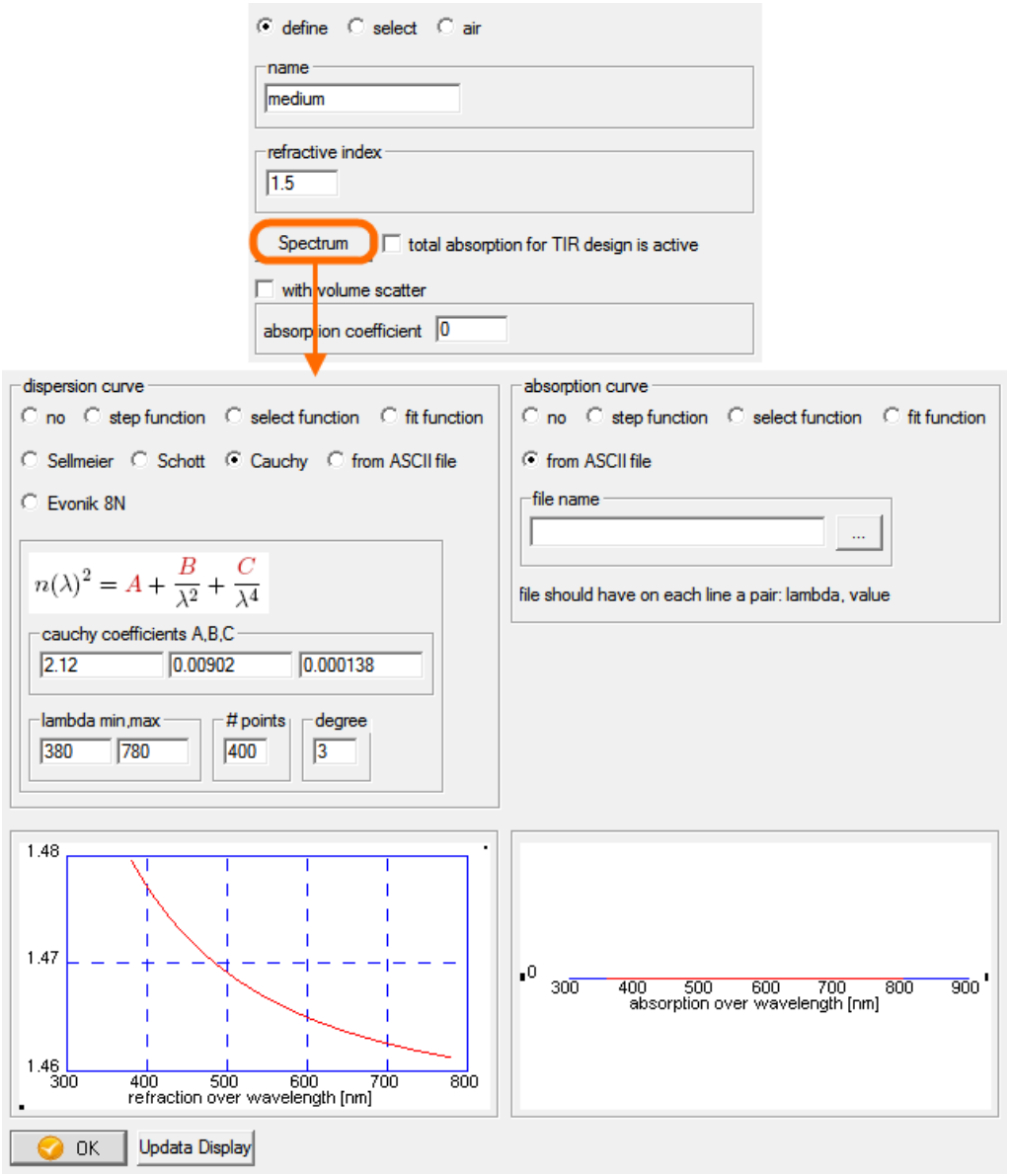


Figure 7.25: The Create Medium Dialog, the Control curves for dispersion and absorption can be set independently

spectral absorption distribution	This is used in spectral simulations instead of the average absorption coefficient. See Section 7.6, “Spectral Reflection, Refraction and Absorption Distributions”.
select from library	LucidShape provides a media library, within several media for glass and plastic are available. This way a special material simply can be added by selecting it in the respective tree-like list.

7.6. Spectral Reflection, Refraction and Absorption Distributions

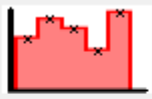
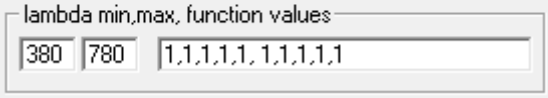
Spectral distributions describe physical reflective, refractive or absorber behaviour of materials or media as a function of the wavelength:

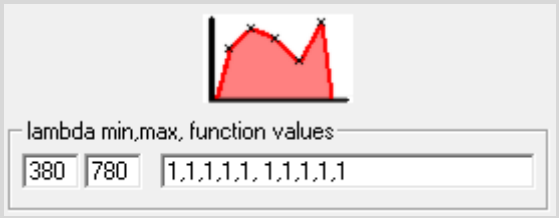
- For a reflector, the spectral *reflection* distribution returns the reflectivity (between 0 = total absorption and 1 = full reflection) as a function of the wavelength.
- For a refractor, the spectral *refraction* distribution returns the refraction index (determining the deflection of the ray when it enters the medium) as a function of the wavelength.
- For a refractor, the spectral *absorption* distribution returns the exponential constant determining the loss of energy of a ray on its way through the medium.

Spectral reflection distributions are available for reflective actor materials, see Section 7.3, “Actor Materials”. Spectral refraction and absorption distributions are available for media (which are in turn used for refractive actor materials), see Section 7.5, “Media for Refractive Materials”. For a more thorough discussion of this topic, please consult the LucidShape Technical Reference.

7.6.1. User Defined Spectral Distributions

User-defined spectral Distributions are specified via the following dialog:

no	No specific distribution is used, which means that the chosen reflectance, refraction index or exponential absorption constant is the same for all wavelengths. This choice yields no further dialog elements.
step function	Defines the distribution function as a step function of arbitrary values which are arranged equidistantly in wavelength. (Please refer also to 'LucidTechnical.pdf'). The values for λ_{min}, λ_{max} set the wavelength interval in [nm]. In detail, the number of entered function values defines the number of steps, where each step takes a wavelength interval of $(\lambda_{max} - \lambda_{min})/N$.  
fit function	The usage of this function is very similar to that of the step function, except that the function values are linearly interpolated to build a piecewise linear function. Here, N+1 values are needed to have N linear segments.



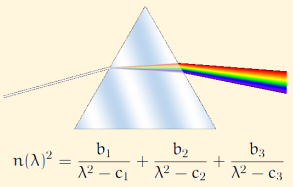
select function	A control curve can be selected from the geo view or tree view. This curve serves as the reflection, refraction or absorption function. The control curve itself contains a list of U and V values.

7.6.2. Special Dispersion and Emission Functions

Besides the option to use any arbitrary user-defined spectral distribution, LucidShape already contains some important special built-in distributions. These are explained in the following sections.

Sellmeier Dispersion Function

Creates a dispersion function according to the Sellmeier equation. This is an empirical relationship between the refractive index n and the wavelength lambda for a particular transparent medium. It determines the dispersion of light in a refracting medium. The below displayed equation is used for glasses. It is created in form of a ControlCurve.



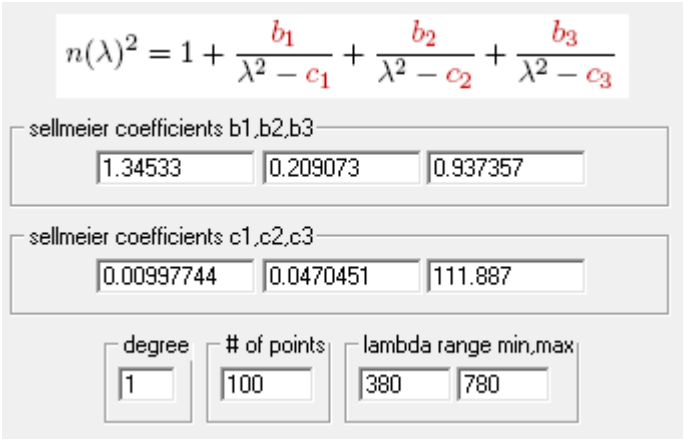
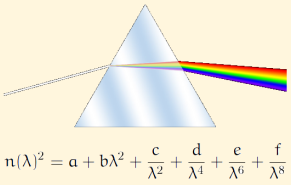


Figure 7.26: The Sellmeier Function Dialog

Sellmeier coefficients	The Sellmeier coefficients, which are experimentally determined.
degree	Determines the degree i.e. the order of the fitting function.
# of points	Determines the number of points the fitting curve consists of.
lambda range min/max	Determines the wavelength interval to be used [λ_{min},λ_{max}].

Schott Dispersion Function

Another form to calculate the dispersion of a material is the Schott equation, which is especially used for Schott glasses. The curve is created as a ControlCurve.



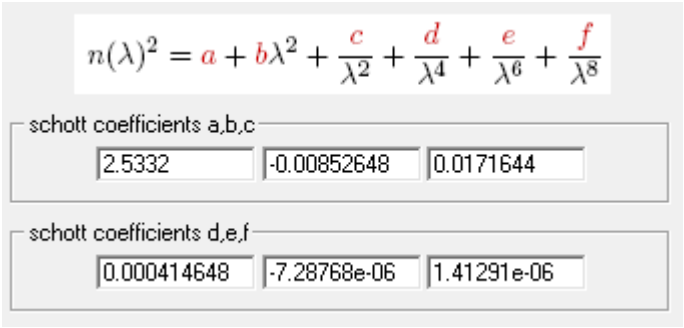
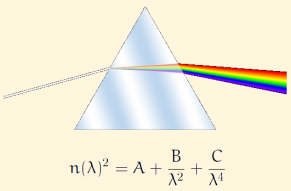


Figure 7.27: The Schott Function Dialog

Schott coefficients a-f	The experimentally determined coefficients for the Schott equation.

Cauchy Dispersion Function

The Cauchy-dispersion function is a special form of the Schott-equation above. Only three coefficients for the exponents 0, 2 and 4 are used the rest is omitted. The curve internally is created as a control curve.



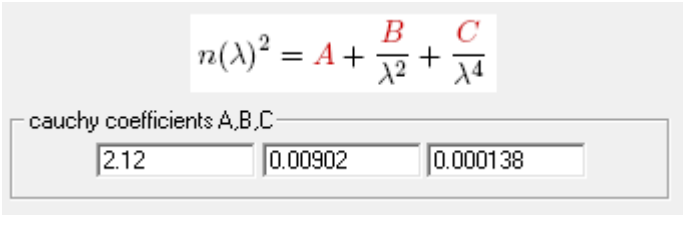
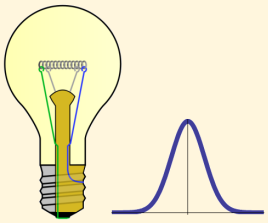


Figure 7.28: The Cauchy Function Dialog

Cauchy coefficients A,B,C	The (experimentally determined) coefficients for the Cauchy equation.

Gauss Emission Function

A Gauss emission function, scaled in such a way that its slope is controlled via the FWHM value (full width at half maximum).



$$f(\lambda) = \exp\left[-\ln(2) \cdot \frac{(\lambda - a)^2}{(\frac{w}{2})^2}\right]$$

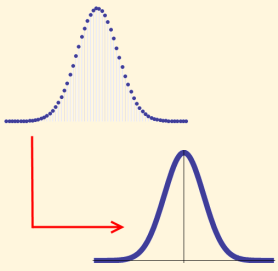
peak (a): 555 full width at half max. (w): 20

Figure 7.29: The Gauss Emission Function Dialog

peak (a)	The location of the peak, has to be located inside the wavelength range.
full width at half maximum (w)	Specifies the width of the control curve. FWHM is the distance between the two points where the curve is down to half of its maximum.

Fit Spectral Curve from Values

This dialog is used to generate spectral distributions from a list of spectral values given by the user. Depending on the selected type of curve (emission, refraction, absorption or transmission), typical initial values are supplied. The transmission curves take further parameters into account to generate the spectral curves.



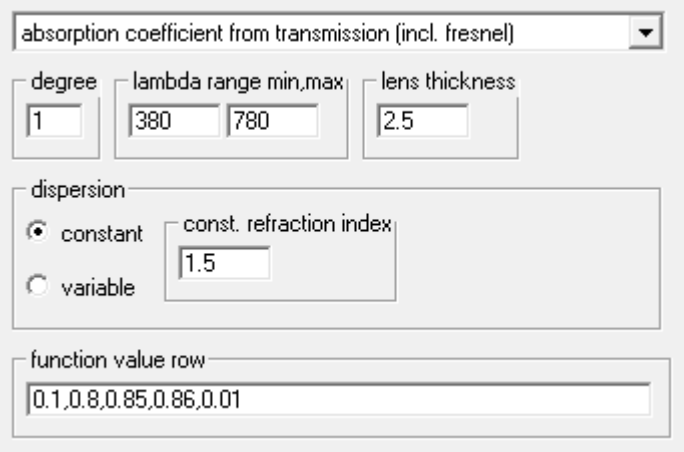


Figure 7.30: The Fit Spectral Curve From Values dialog

pull-down menu: curve type	Determines the basic setup of the fit curve, available choices are: • emission • reflection • dispersion • absorption coefficient • absorption coefficient from inner transmission (no fresnel) • absorption coefficient from transmission (incl. fresnel) Depending on this choice, the function value row will be preset by the program.
function value row	The intensity values within the lambda range. The number of values define the discretization of the curve.
lens thickness	Parameter which is only available for the two choices *absorption coefficient from inner transmission*.
dispersion	Parameter which is only available for the choice *absorption coefficient from inner transmission (incl. fresnel)*. May be supplied as a constant or as a spectral dispersion curve.

7.7. The BSDF Wizard

7.7.1. BSDFs in a nutshell

Recall that BSDFs (bidirectional scattering distribution functions) are used to specify diffusely reflective or refractive surfaces. In LucidShape, BSDFs are realized by control curves.

For getting a rough understanding of how BSDFs model the scattering behaviour of a reflective surface, first consider the classic mirroring surface (see picture right below): An incoming ray is reflected by the surface, the outgoing ray's direction is determined by the classic reflection laws from the incoming ray's direction and the surface normal. No scattering of light into adjacent directions occurs, so this classic (non-)scattering behaviour is described by a BSDF which is 1 at $\delta=0$ and 0 for $\delta\neq0$. The angle δ stands for the deviation from the direction of specular (ideal) reflection.

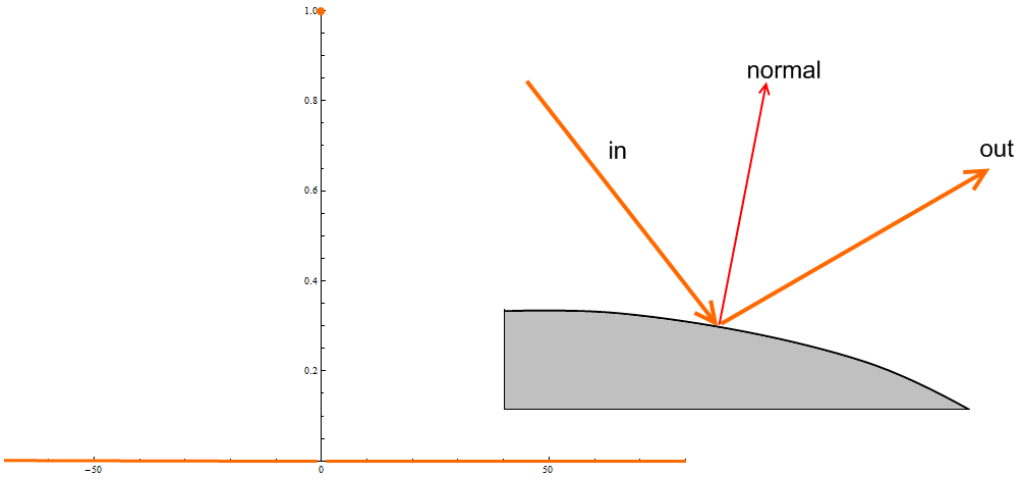


Figure 7.31: Classic reflection without any scattering: The BSDF=1 at $\delta=0$ and BSDF=0 at all other values of $\delta\neq0$.

In contrast, Lambertian scattering is described by cosine BSDFs and Gaussian scattering (not surprisingly) by Gaussian BSDFs (see picture right below).

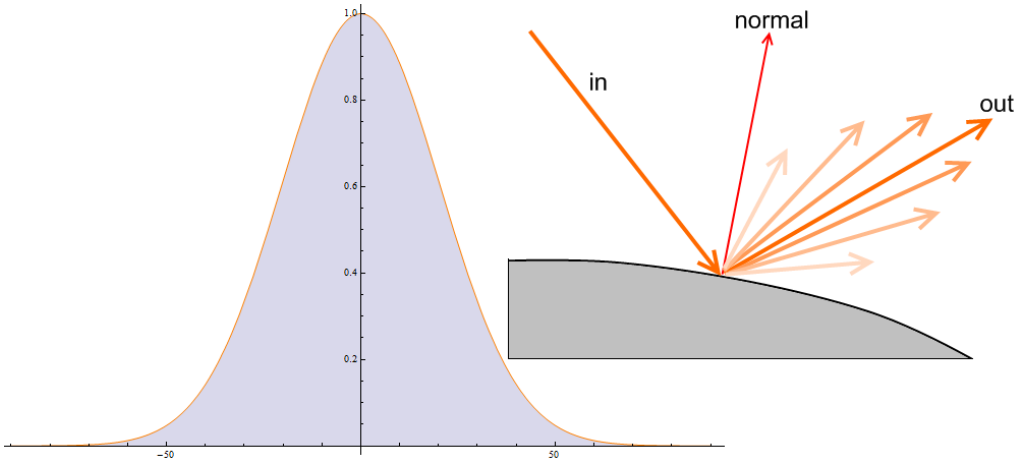


Figure 7.32: Gaussian scattering: BSDF is a Gaussian Curve.

More general, BSDFs are often based on numerical measurements of a material, taken from different light input angles. Usually the data is stored in tabular form. Depending on the respective supplier, it can be saved e.g. in three columns, containing the angle pair theta and phi and the belonging intensity. In other cases there is a separate file for each theta angle, so the measured data is not stored in a single file but in a folder.

Here comes the LucidShape BSDF wizard into play: It reads complete data sets gathered from such samplings and transforms them to suitable BSDF curves.

7.7.2. The Wizard's Dialog Elements

Basically, the BSDF Wizard consists of the following four parts.

- File input part: BSDF data files may be read in several formats. Usually, multiple files are read where each file contains a material measurement from a fixed input angle. The data files appear in a file list.
- Curve type creation part: Select here the types of BSDF curves to be created.
- Filtering part: Enables the smoothing of BSDF data.
- Abscissas axis type: Sets the axis type for the BSDF curve's display in the geometry view.

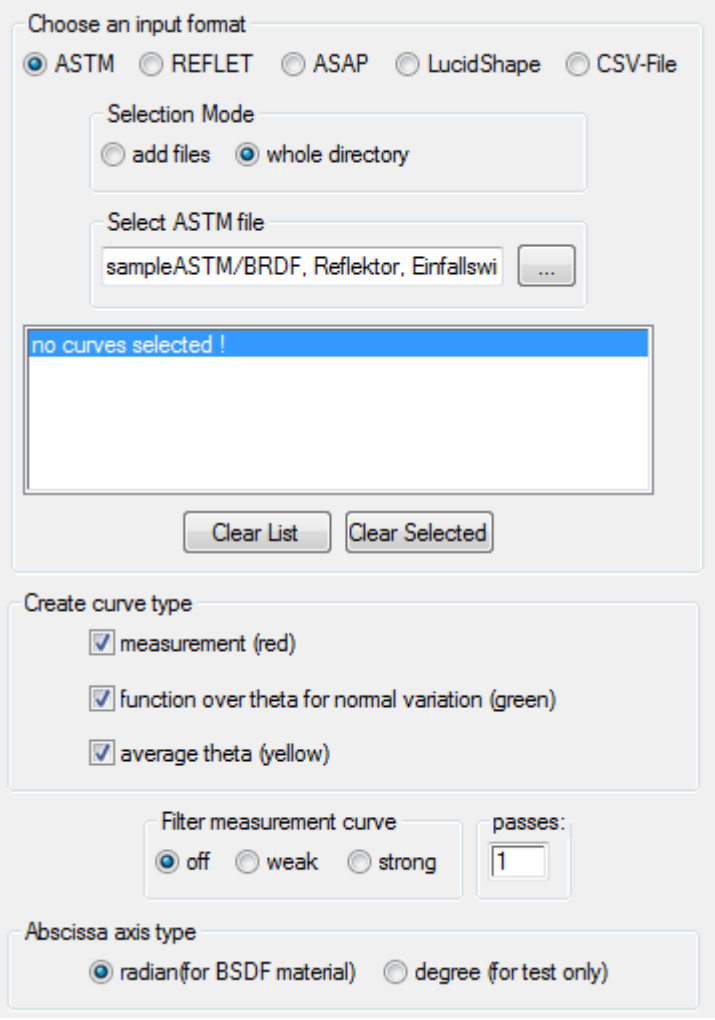


Figure 7.33: BSDF Wizard Dialog

radio button: input format	Sets the file format for the BSDF data import. Available formats are: ASTM, REFLET, ASAP, LucidShape and .csv files..
radio button: selection mode	Determines the mode how files are selected by the user to appear in the file list. If *add files* is selected, single data files may be entered one by one. If *whole directory* is selected, a directory may be entered and all of the files in the directory will be added to the file list.
inclination angles (LucidShape and .csv)	The inclination angles can be used to add a constant shift onto the input angles. The number of values to be entered has to match the number of files in the list, the values separated by a blank character.
select file	The file or directory selection dialog.
button: clear list	Removes all files from the current list.
button: clear selected	Removes the currently selected file from the list.
flag: measurement (red)	If checked, for each data file in the input list a control curve is generated, each one is centered around the particular data set's input angle.
flag: Function over θ normal variation (green)	If checked, for each data file in the input file list a control curve is generated from the measurement curves via normal variation. These curves are centered around the o degree axis.
flag: Average θ (yellow)	If checked, computes one BSDF control curve from the previously described curves. This curve may be used in subsequent steps to construct actor materials.
radio button: Filter measurement curve	Determines the type of filtering. Available are no filtering at all, weak and strong filtering. The choices weak and strong enable one-dimensional smoothing of measurement data on their input into LucidShape. Additional the number of smoothing passes (default: 1) may be specified.
radio button: Abscissae axis type	The curves are created either with radians as their domain units (horizontal axis) or with degrees as their domain units (horizontal axis). This is for test purposes only and not to be used for "real" BSDF curves.

7.8. Importing other BSDF Data

Beside the BSDF wizard other BSDF Data easily can be imported just by drag & drop (among other .brdf, .mac or .ref). The header of the respective file is scanned and the belonging dialog opens up for performing the last adjustments.

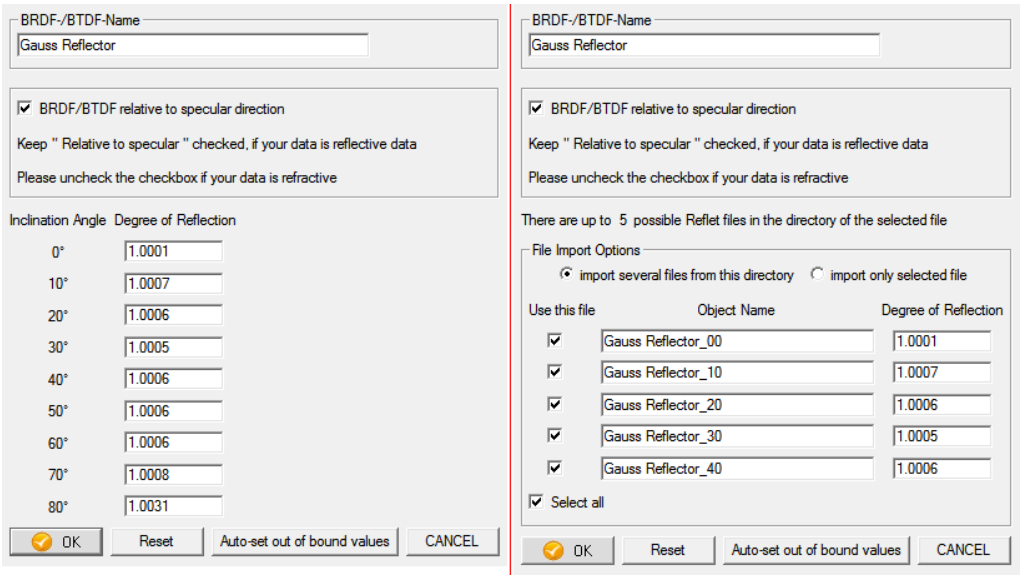


Figure 7.34: BSDF Import

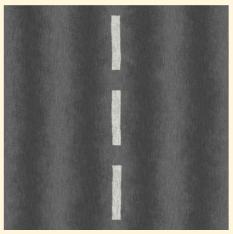
name	The name of the BSDF Data to appear in the TreeView.
flag: relative to specular direction	Depending on whether the data is BRDF or BTDF, the flag has to be checked or unchecked.
button: auto-set of bound values	When importing the file(s), among other the *Degree of Reflection* (DoR) is computed. In case one value (of an angle or a file) is outside the range [o,1], the measurement cannot be imported. One can either re-scale this single file (by manually changing the value) or the complete measurement by pressing this button.
radio button: file import options (REFLET)	In case of REFLET data the single files do not contain the complete measurement, but single angles. With the radio button the user can choose whether the single angle or the complete data shall be imported.
flag: use this file (REFLET)	This flag can be used to in- or exclude single files from the export.
flag: select all (REFLET)	If checked, all listed files are taken into account.

7.9. Set Colour

Sets the colour of objects (either all, only selected or for selected object vectors).

7.10. Set Texture

Sets a texture on the currently selected surfaces. The texture bitmap file may be *Display Texture* from the context menu in the geometry view.



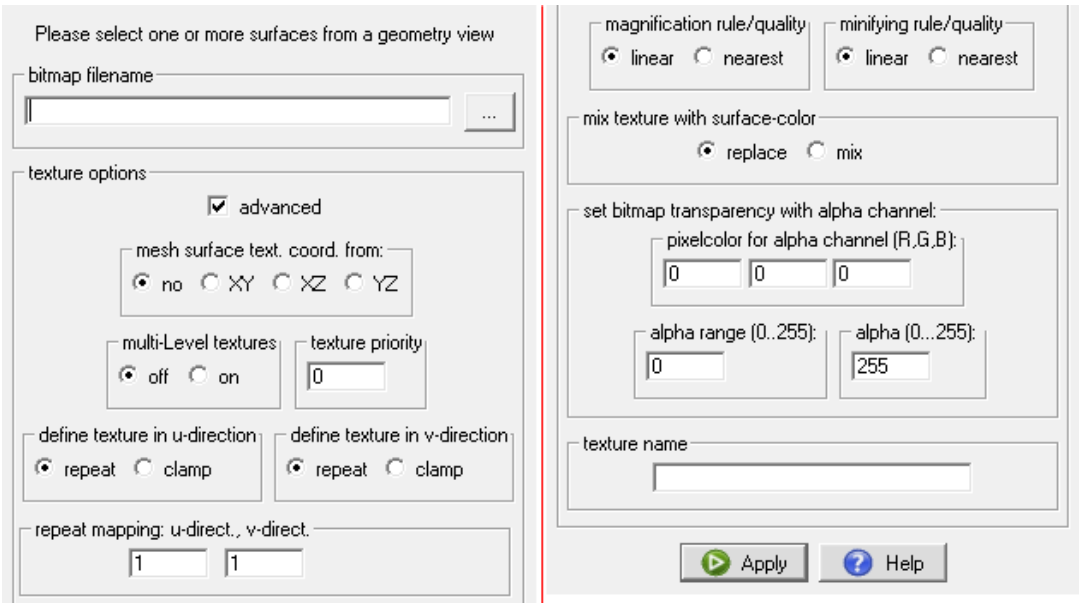


Figure 7.35: The Texture dialog

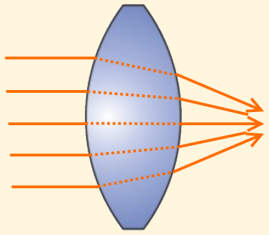
bitmap filename	After an object has been selected, one is able to choose a picture (in various formats) and set it as a texture onto this object.
flag: advanced texture options	This flag enables some further options and parameters.

radio button: mesh surface texture coordinates from	Specifies the plane on which the texture shall be set (*no* is kind of an automatic detection, the other three possibilities set it directly).
radio button: multi-level textures	This en-/disables MIP maps, for different distances to the object different resolutions of one texture are used to save memory and time for computing (for a great distance to the object a lower resolution is used, for a very close view the highest resolution is used).
texture priority	Sets a priority (from 0 to 1), this determines the handling of the textures inside the graphics (if there is not enough memory for all the textures).
define texture in u-/v-direction	If the picture is smaller than the object, it is possible to just take it that small or repeat it until the whole object is covered.
repeat mapping in u-/v-direction	Divides the object into the entered number of areas, the texture is set onto each of these areas.
radio button: magnification/minifying rule/quality	Select the type of how to resize to either linear (standard method) or nearest neighbour (slightly better quality).
radio button: mix with surface-color	The object itself has a color, with this option one either completely replaces this object color or allows a mixture of object and texture.
pixel-color	Defines the exact color which shall be modified via the *alpha* value below.
alpha range	Enables an interval around the exact pixel-color so that the alpha of more than one single color can be modified.
alpha	This value sets the transparency of the texture, the range goes from 0 (complete transparent) to 255 (in-transparent).
texture name	Set a name for this new texture to appear in the TreeView.

Chapter 8. Simulation

8.1. Monte Carlo Ray Trace

The so-called *Monte Carlo ray trace* is the most important simulation method for a given emitter+actor+sensor setup. The basic algorithm is straightforward (but nevertheless computationally complex): Rays are computed randomly from the emitters (=light sources), they possibly interact with the actors (=reflectors or lenses) to finally reach the sensors (=detectors) where they are accounted. This type of simulation is accurate and flexible, because it can handle all kinds of optical setups, possibly containing hundreds or thousands of reflectors and refractors.



In LucidShape, Monte Carlo ray trace is implemented in various flavors. You may directly start a simulation for your model in focus, or you can relocate the simulation to a number of external parallel processes. You can even handle all the computations in a spooler that is capable of distributing computational loads to individual computers in your local network.

There are a few common messages which are shown in the MessageView after the simulation process has finished. They give some information about the behaviour of the rays and what happened to them (e.g. they have been stopped because of a too low intensity, reached the maximal recursion level, or similar). Please refer to the section called “Message View”.

*number of rays *1000*	Determines the number of rays to be used for the simulation, e.g. '10' results in 10000 rays. The default value is 100^5, which is a reasonable number for obtaining a very first sketch of the "real" simulation result. Depending on the complexity of your model and the resolution of your sensors (the cell-size), an estimated 10^7 to 10^{10} rays are necessary to obtain a realistic result. The computing time increases linearly with the number of rays.
button: set max	If pressed the maximum possible number of rays is entered. While using a ray file the total number of rays in this file is applied, otherwise there is only a limit depending on your used system.
# loops	Determines the number of simulation parts the complete simulation is divided into. After each part has been computed, the sensors are updated with the respective preliminary result, so one can have a preview to the LID without having to wait until the complete process is done.
recursion	Determines the number of surface interactions a ray may have through the scene before it is considered as 'trapped' and thus abandoned. This is helpful to prevent program deadlocks produced by infinite reflections (e.g. a ray inside a mirrored sphere would be reflected an infinite number of times, if it is not abandoned). While using shapes like e.g. prism band, a higher number is recommended due to the special behaviour of this shapes.
flag: spectral simulation	If checked the spectral simulation is enabled. The available parameters are the number of *spectral channels* and the spectral

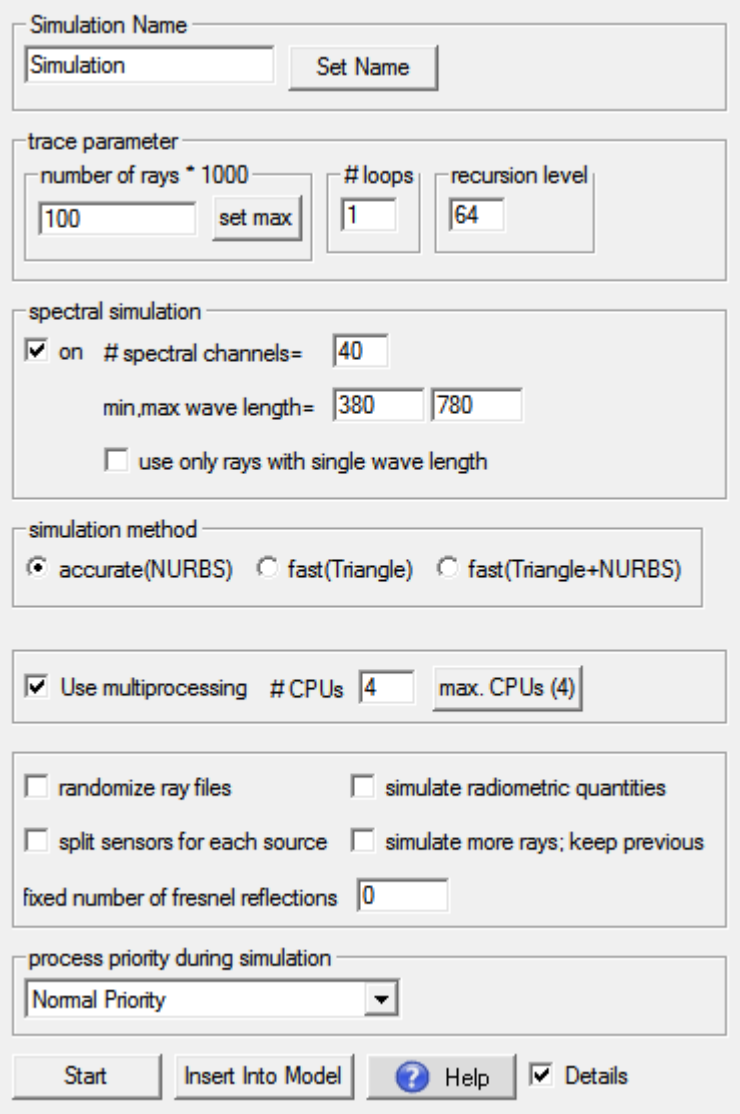


Figure 8.1: The Monte Carlo Ray Trace Dialog

	wavelength range. The wavelength range determines the range of the complete used spectrum, the channels divide this spectrum into evenly divided smaller parts. (For more detailed information please see also Chapter 3 and 5 of LucidTechnical.pdf.)
flag: use only rays with single wavelength	Regardless if this flag is en- or disabled, the simulation result contains spectral information, the only difference is the effort during a simulation. If checked, LucidShape is dicing the wavelength of each starting ray (the wavelength distribution is due to the SPD defined) and this wavelength will be counted in the sensor. If unchecked, one ray carries the complete spectral information, i.e. the spectrum defined by the SPD and this complete spectral distribution will be counted in the sensor. When such a ray intersects with a wave length dependant surface (i.e. a surface with a

	dispersion or wave length dependant absorption), LucidShape is dicing a wavelength and from this intersection coordinates on, the ray is traced using this single wavelength. The second case has a higher effort during a simulation. So when there are wave length dependent surfaces in the model, it is recommended to activate the flag.
radio button: simulation method	• NURBS: This is the most accurate simulation method because it directly uses the mathematical parametrization of the shape's surface. Du to the calculation process, in general this is the most time-intensive simulation method. • Triangle: Here the shape's surface is recreated by triangles, their size is determined by the given tessellation parameters (see below). The ray-tracer starts a ray from the emitter and computes the intersection coordinates, i.e. which of the triangles is hit by it. Next the ray is deflected due to the material of this shape, using the normal of the hit triangle. • Mixed: This is the combination of both NURBS and Triangles, here the surface is recreated by triangles too. Again the ray-tracer computes the triangle which is hit. In contrast to the method *Triangle*, the intersection coordinates are not taken from the triangles' surface, here this point is projected back onto the original shape's surface. This slightly increases both the accuracy and the simulation time needed for this method.
tessellation parameter	There are not THE values one can enter for a 'perfect' i.e. correct simulation. Before performing a simulation, always press *Apply* to see the effects of the current settings in the GeoView! In case the surfaces do look deformed or strange, try adjusting the values in the dialog. Perhaps the maximal number of auxiliary nodes has to be increased → Section 12.2.1, “Global Settings”. • minimal edge length: This is the minimal length of a triangle's side. This lower limit is needed to prevent too small triangles. • maximal edge length: This is the maximal allowed length of a triangle's side. Lowering this value will force the tessellation process to create more triangles, the recreated surface becomes 'smoother'. • max. chord height: This is the maximal allowed distance between the original surface and the triangle created here (also see the sketch below this table).
flag: use multiprocessing	If checked the multiprocessing is activated and the number of CPUs can be set, highly recommended if not more than one instance of LucidShape is running on your system at the same time. Pressing the button 'max. CPUs' will set the entered number to the greatest possible value (as stated inside the brackets).
flag: randomize ray files	When performing a simulation with less rays than available in the ray file, this flag can be used to randomly choose among the single rays. This is important because some ray files are created in a

	certain order, e.g. the resulting LID is building up in (anti-)clockwise direction or the ray file is sorted with ascending theta angle.
flag: split sensors for each source	If checked a sensor is created for each source. After the simulation has been completed, right-click into the LID and choose *navigate in 3D/4D data set*, here one can analyze the result for each source of its own and the sum of all sources together.
flag: simulate radiometric quantities	If checked the simulation will be done with radiometric ([W]) instead of photometric quantities ([lm]). Note that spectral simulations with photometric quantities are scaled with a so called V-lambda curve, to account for the reception characteristics of the human eye. This choice determines the physical unit stored in the resulting RayFile when using a ray file sensor to record the simulated rays.
flag: simulate more rays, keep previous	If checked the sensors will not be cleared when starting the next simulation (i.e. deleting the stored data) but will keep the already simulated rays, so the next simulation will be added to the already obtained data (this is kind of a manual version of simulating with loops).
fixed number of fresnel reflections	Usually only a very small percentage of the rays intersecting with a refractive surface are reflected due to the fresnel equations. To increase this percentage significantly, this parameter was added to ray trace dialog. A value of '0' means that the simulation takes place as usual, i.e. only a small amount of rays is reflected and the majority of the rays are transmitted, the ratio of these two amounts is determined by the two media the surface is separating. A value of '1' will result in this new behaviour: The rays are starting at the emitter as usual. When intersecting the first surface, 50% of them will be reflected, 50% will be transmitted (this is a big difference to the usual ~4% reflected by standard window glass). The intensity of the rays is weighted that the resulting amount of light in the sensor ([lm], [lx], etc.) is exactly the same as when using '0'. A value of '2' means that this special procedure will be performed for the first two surfaces hit by the ray, etc. The intended effect is to increase the amount of (fresnel-)reflected rays significantly, so the noise in the sensor is reduced significantly, too. This might very helpful when using the Luminance Camera.
process priority during simulation	Sets the priority under which the simulation process runs on your Windows system. If set too high, this may render your system unresponsive. It is strongly recommended to keep the setting *normal priority*!
flag: details	If unchecked the dialog changes to a minimum version, the only accessible parameter is the number of rays.

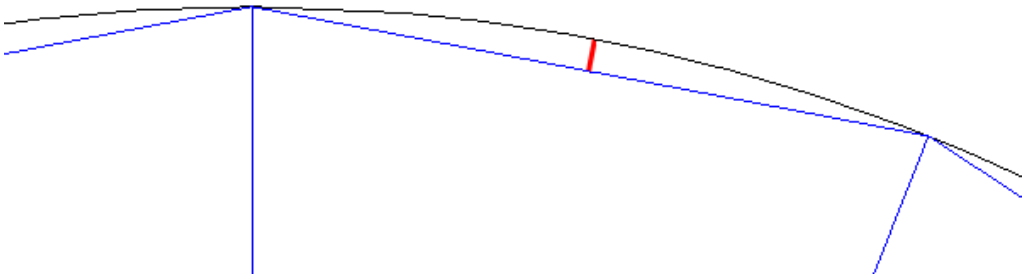
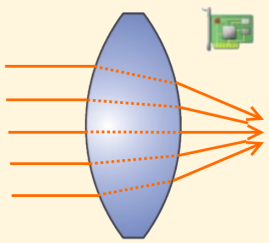


Figure 8.2: 2D Sketch with activated Tessellation: The original round surface is drawn in black, the triangles in blue, the distance (red line) is the chord height.

8.2. GPU Trace Simulation

This method is quite the same as the Monte Carlo Simulation as it uses the same parameters and options.

The major advantage is the additional use of the graphic card, some parts of the computing are transferred to the GPU, where the calculation is magnitudes as fast as on a common CPU due to the special architecture of the GPU (keyword "massively parallel computing").



Please note: The GPUtrace Simulation is only available for nVIDIA® graphic cards because CUDA is needed for the calculation process. Please also refer to http://www.brandenburg-gmbh.de/products/gputrace/requirements/ for further informations.

Furthermore, there is no possibility to select the simulation method NURBS, the only available type is *Triangle*. Due to the internal programming, the RayHistorySensor and Volume Scattering are not available when using the GPUtrace.

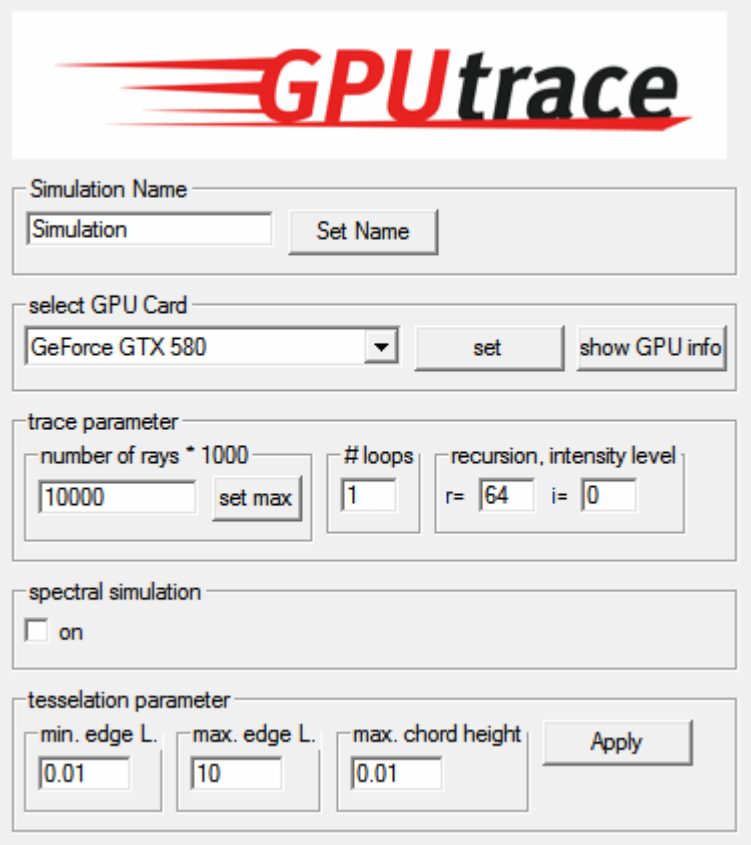
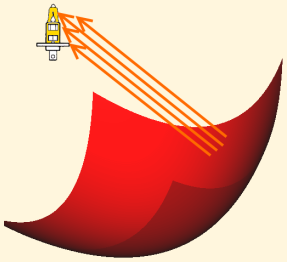


Figure 8.3: The GPU Trace Simulation dialog

The drop-down menu *select GPU card* allows to select the GPU which shall perform the ray-trace. This choice is independent to the rest of the system (e.g. the GPU used for the monitor), but of course the less additional work the chosen GPU has to do, the faster the simulation is done.

8.3. Light Mapping Simulation

In contrast to Monte Carlo simulations (which intersect rays directly with actor shapes and thus require an enormous computational effort for the ray-surface-intersections), the *Light Mapping* simulation method starts directly on the actor surfaces and gets light rays from the sources that reach the position on the surface. This way, the ray-surface-intersection points do not have to be computed because they are already given as starting points of the method.



As a result, the light mapping simulation is significantly (sometimes by orders of magnitudes) faster than Monte Carlo simulations. Major drawbacks of light mapping simulations are their limitation to pure reflector models (the trick of getting rays from the source does not work for refractive shapes) and the dependence on the parameterizations of the reflector surfaces (bad parameterizations make balanced distributions of points on the surfaces difficult, which in turn leads to unbalanced and thus inaccurate simulation results). Also, light mapping is not capable of accounting for

stray light which comes directly from the sources. For many pure reflector setups, however, light mapping will give a reasonably accurate and very fast impression of the final result.

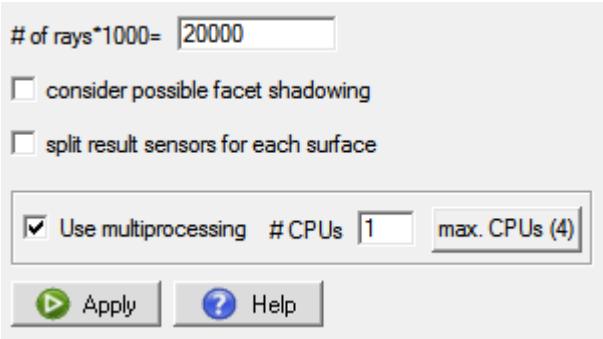


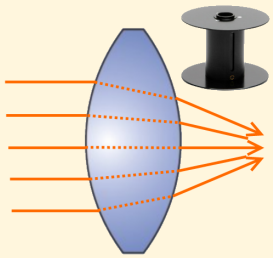
Figure 8.4: The Light Mapping Dialog

*# of rays *1000*	The number of rays which should be used for the simulation.
consider possible facet shadowing	Standard light mapping of a reflector surface in a model does not detect shadowing effects (for example, if another surface absorbs parts of light rays from the source). If this flag is enabled, the light mapping is enhanced in such a way that it checks for such situations. Note that this slows down the simulation.
split result sensors for each surface	The result sensors are split into separate channels according to the number of surfaces. The first channel holds the complete simulation result of all surfaces and each following channel holds only the result of the single surfaces. These light distributions can be examined using *Navigate through UV Blocks* under the UV-data's context menu.

8.4. Simulate by Spooler

The LucidShape simulation spooler carries the idea of simulating in external processes another step further and allows the simulation of your experiments on remote computers in a local network.

A very important point: Make sure the created spooler directory can be accessed and edited from the network (keyword 'writing authorization')!



Roughly spoken, the spooler can be used in three more or less different ways. The most simple way is to use the it as a local spooler on one PC. The user does the setup, starts the spooler and 'feeds' it with simulations jobs.

The second possibility is to setup the root path in the local network so another PC can access the jobs (e.g. a remote-controlled one via TeamViewer or similar). On both PCs the spooler points at the same directory. On the remote-controlled PC the spooler is started. i.e. it waits for work jobs

arriving in this root path, these jobs are created by the local PC. This way one can prepare simulation jobs on a non-busy PC while another PC is performing the simulations.

The third possibility is to setup a real Server/Client spooler. Here the local PC becomes the Server, and more than one PCs are assigned as Clients.

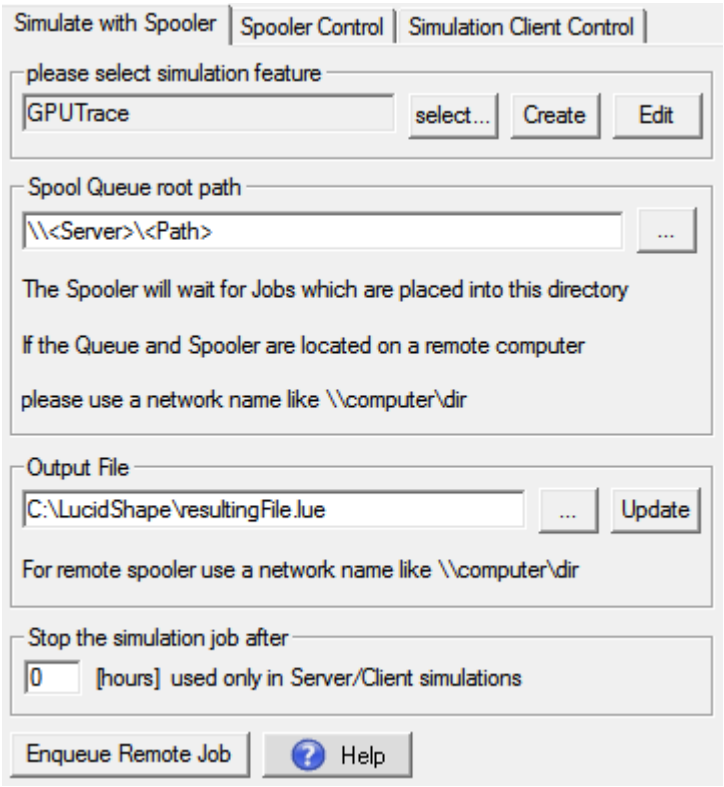


Figure 8.5: Spooler Dialog

For a local spooler the initiation sequence and the further handling works as follows:

- Open a file explorer and create a new *spool queue root path* or select an existing path via the button '...'.
- Open the model you want to simulate and create or select a simulation feature.
- The *output file* shows the location and name where the file is saved after the simulation has been performed. Pressing *Update* will show the default path and name, both can be altered.
- Pressing *enqueue remote job* will add this current job to the spool queue. More precise a small script file is created which will trigger the real simulation when called.
- After all jobs have been added to the queue, switch to the tab *spooler control*.
- Pressing *list* will list all jobs which are currently in the queue.
- The buttons *start* and *stop* will start/stop the spooler, i.e. the local simulation of the jobs.
- Pressing *clear* will clear all jobs enqueued.

This procedure is nearly identical for the second possibility. The two paths specified have to be identical, but the spooler is started only on the remote-controlled PC. The local PC is used only for adding jobs to the spool queue.

The third setup is a bit different and works as follows:

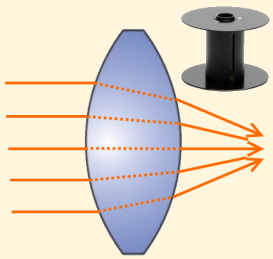
- On the local PC to become the Server, open the file .\samples\spooler\startspooler.bat in a text editor (but not with Windows notepad.exe). Change the string 'spooldir' into the spool path to be created (this is the spool root queue path already specified + the subfolder '\spoolQueue'). The result should look similar to this: \\‹SERVER›\‹path›\Spooler\spoolQueue
- Replace the string 'rlush' with the complete path of the lush.exe (this .bat script is a windows script, so it has no access to the paths specified in the LucidShape.ini file).
- Open a command window in the directory of the startspooler.bat and execute this .bat file within.
- Now switch to the first PC to become a Client. Enter the name of the Sever PC into *Spool Server Computer* and press *start client*. Repeat this for every PC which shall contribute.
- Now the setup has been finished and simulation jobs can be added from the Server PC.

After the initialization of the spooler has been done and the first job is enqueued, there are two temporary folders created below .\Spooler\, after all simulations are done and the queue is empty, these two sub-folders may be deleted.

Please note: Due to security reasons the assignment of clients to the computer network may/can not be done from the Server computer.

8.5. Simulation Manager

The Simulation Manager is a bit similar to the spooler, but with a few differences. Jobs are not added on the fly from a currently open model, the jobs are added as saved .lug files.



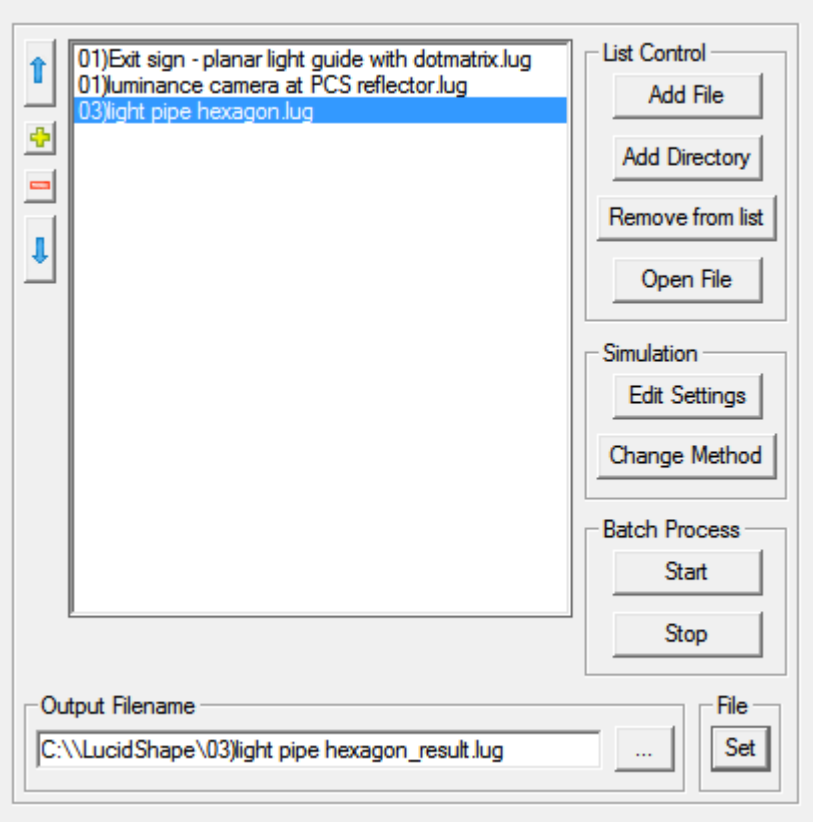
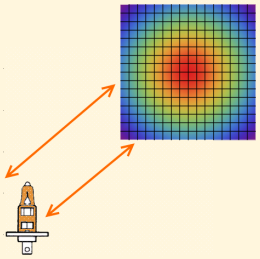


Figure 8.6: The Simulation Manager Dialog

arrow buttons	These two arrow buttons can be used to manually sort the order of the simulation jobs.
button: add file/+	Press one of these two buttons to add a .lug file to the job list.
button: add directory	Press this button to add a complete directory to the job list.
button: remove from list/-	Press one of these two buttons to delete the currently selected entry.
button: edit settings	If pressed, the simulation dialog opens up.
button: change method	If pressed, one can select the simulation method.
button: start	Pressing this button will start the Simulation Manager, starting with the first entry in the job list.
button: stop	Pressing this button will stop the ray-trace process. The resulting file is not written, but the job is still in the list.
button: ...	Press this button to select the path where the resulting file shall be saved.
button: set	One can alter the path and the file name of the result directly in the input field. Pressing this button will apply the changes.

8.6. Sensor Light Calculations

The sensor light calculation commands use a different approach on collecting sensor data. Instead of starting at the light-source and following the paths of different rays, these methods focus on the sensor and track rays directly back to the source. Thus, they will neglect optical components like mirrors.



8.6.1. Gather Sensor Light

Performs a sensor light gathering, this command has no parameters or options at all.

This method may be applied for a scene where all sensor objects are at a far distance to the light source with respect to the light source's size. The light in each sensor is directly gathered from each light source. No effects like reflection at other objects are taken into account. So there is no contribution of optical components like mirrors. The sensor will "see" only the light-source (emitter), but shielding by other objects (in between) is considered.

8.6.2. Reverse Sensor Light

In most cases one has a model with some actors, at least one sensor and one or more emitter (however they have been created). Next a ray-trace is started to achieve a resulting LID. This is done by simulating the interaction of the generated rays through the model (propagation, intersection, absorption, etc.).

But there are also some cases the opposite way is wanted. Here one has a LID (however it has been created) and is looking for an emitter which is able to reproduce this present LID. To reproduce means the calculated emitter is simulated in an usual forward ray-trace and as a result we achieve the same LID we already had before the complete procedure was started. The calculation process can be done in LucidShape and is called *Reverse Sensor Light*.

This type of reverse ray-trace is used in some of the applications listed in Chapter 11, *Applications*.

The reverse process is done as follows: First, you define a virtual point light source. The light in each sensor of your setup is projected back into that virtual point source. The light in the virtual point light source is stored in a light data object and displayed in a data view.

Reverse sensor light calculates a luminous intensity distribution from given sensor data as if that distribution had been emitted from the virtual point light source. From a previously simulated scene with light in all sensors or with a scene set up "by hand", we can calculate this reverse sensor light. The obtained result is a LID. This LID will reproduce the former light distribution in all sensors, when it is illuminating the original scene instead of the original light source. There is a conceptual affinity to holography.

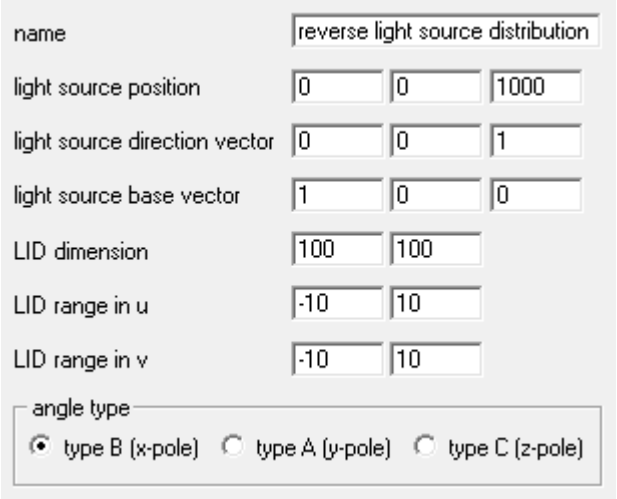


Figure 8.7: The Reverse Sensor Light Dialog

name	The name of the reverse sensor light distribution.
light source position, light source direction and base vector	The position and the orientation of the local axis system of the virtual point light source.
LID dimension	The number of entries in u- and v-direction.
LID range in u, LID range in v	The UV-range of the light intensity distribution, i.e. the cone of interest.
angle type	Determines the angle type, i.e. type B (x-pole), type A (y-pole), or type C (z-pole). See the Technical Reference manual for more detailed information.

8.6.3. Luminance Image Sensor Light

A luminance image is created by a fast backward ray trace where the light is collected from the "loaded" sensors in the scene. The luminance distribution is always seen from a given axis system. Please note: Luminance is an "per area" based dimension. In consequence, only surface sensors are compatible with this function, there has to be at least one sensor for, let's say illuminance [lx].

# of pixel in u,v	This determines the resolution of the image to be created.
flag: create color image with tri-stimulus XYZ luminance	If checked the image is created in the XYZ color format. (This format is the same as in the usual sensor dialogs.)
flag: create N x M panorama cameras	If enabled, a panorama of cameras will be created, similar to the panorama in the luminance camera.
horizontal/vertical range	For both ranges the given number of cameras/pictures are equally distributed in the interval [start angle, end angle].
camera	The view position is the position the camera looks at. Enabling the flag *view at*, the distance between the camera and the viewing plane can be set. The view direction is the direction, the camera looks at the view position. Using the default setting of (0,0,-1) the camera looks in

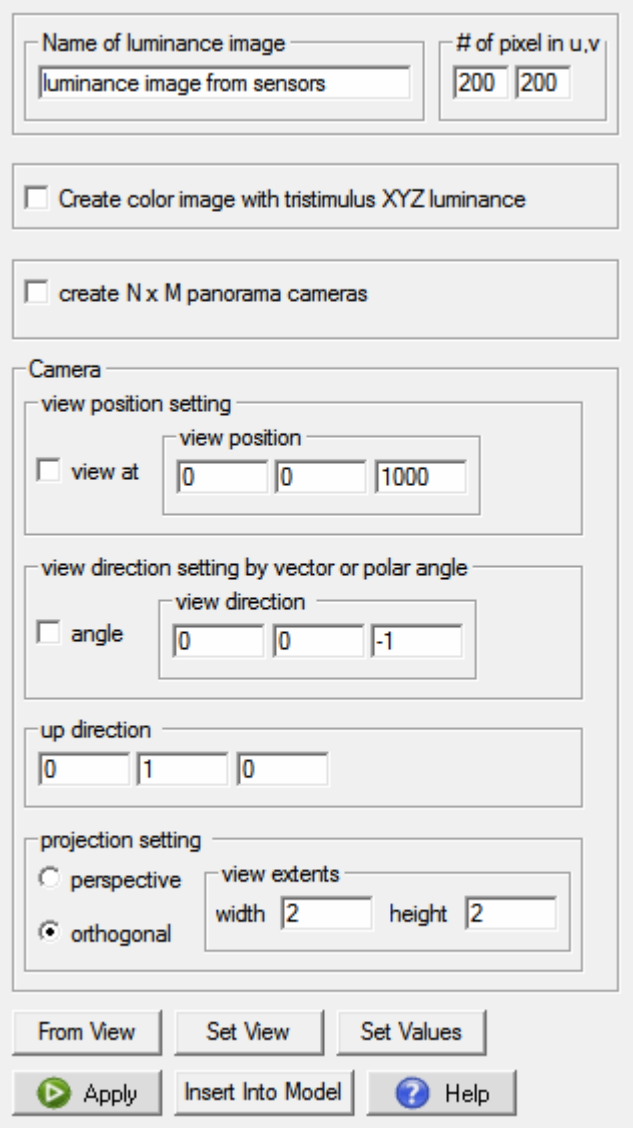


Figure 8.8: The Luminance Image Sensor Light Dialog

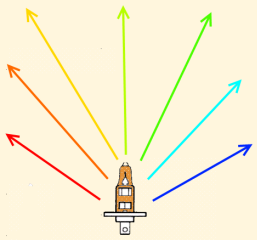
	-z direction. This direction can be specified either in cartesian or polar coordinates. In case of using a panorama, its range is set relative to the view position and direction.
up direction	Determines the orientation of the y-axis of the just specified axis/viewing system.
projection setting	With these parameters the field of view can be set to either perspectively or orthogonal form.
view extents	The view extents determines the size of the visible area the camera looks at. This is analogous to the field of view used in the luminance camera. (Please see also the buttons below in this table.)
button: from/set view	Pressing these two buttons allow to directly exchange the camera parameters with the GeoView. At first one should modify the

	GeoView (position, direction, size) until it shows more or less the wanted picture. Next press *from view*, do some adjustments in the dialog, apply these settings onto the model by pressing *set view* and finally press *apply* to create the luminance image.
button: set values	Applies the values to the model, but does not create the luminance image.
insert into model	If pressed, this feature is added to the model and shown in the TreeView. This entry can be used to open the edit dialog or to delete this feature.

8.7. Various Commands

8.7.1. Random Rays

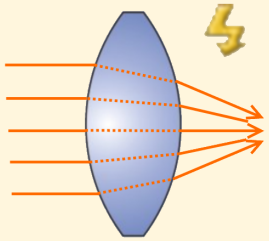
Creates random rays and displays them in the geometry view.



8.7.2. Interrupt

Stops the actual running ray trace, independent of the simulation type like LightMapping, MonteCarlo or GPUtrace.

This interrupt is not applicable for simulations that run in external processes, i.e. spooler simulations.



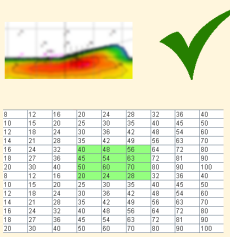
Chapter 9. Analysis of Geometry and LID Data

The tools in this chapter may be found under *Analysis* (for the 3D oriented tools) and *UVAnalysis* (for the UV data oriented tools). Please Remember: According to the type of view in focus, the menu names in LucidStudio will automatically adopt to the appropriate view.

9.1. LID Measuring

9.1.1. Apply Test Tables

A dialog which offers access to a broad selection of test tables for Light Intensity Distributions (LIDs) in UV Data views. LIDs can be checked with the tables to meet legal automotive regulation requirements.



(ES means right handed system / LES means left handed system)

To apply the selected test table, make sure to have the wanted UVView to be checked in focus.

On request, we can extend the test tables for your special needs, we also can add additional tables which are not yet included. Due to the reoccurring changes in legal regulations however, you should check that the test table you need is up to date with the requirements for your country. You are always welcome for any comments on state-of-the-art legal regulations.

For additional informations about the customization of the test tables and a description of the single measurement tasks, please have a look at the new whitepaper *LIDTestTable.pdf* (which can be found at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/whitepapers/) and in Section 12.2.3, “Editing of LID Test Tables”.

Please note that the values in the output window are rounded, so even if a measured value seems to be identical to a legal limit, it may be displayed in blue or red!

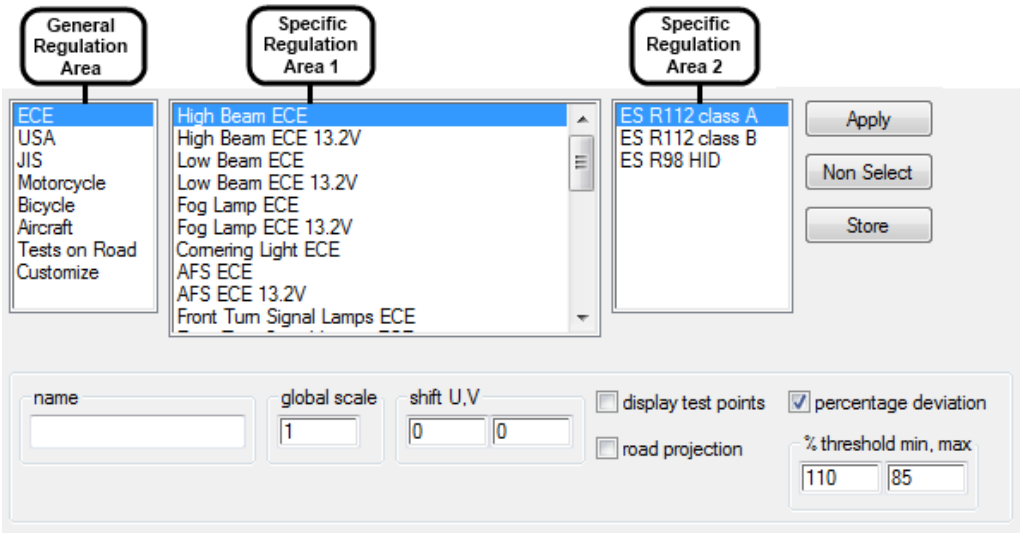


Figure 9.1: The LID Measure Dialog (I)

general regulation area	Select here the general regulation area like e.g. ECE, USA, JIS, etc. The second list and its entries are created automatically due to the selection done here.
specific regulation area 1	Select here the specific regulation area like high beam ECE, low beam ECE, low beam JIS, etc.
specific regulation area 2	Select here the exact regulation for your application.
vehicle length	In some regulations for side-markers, the test table depends on the vehicle length, to be entered in [feet].
global scale	The LID is multiplied with this global scale factor before the measuring is applied. (The use of this factor is similar to the parameter *global scale*, available in sensor dialogs.)
shift u,v	Shifts the light intensity distribution uniformly in the designed u/v-direction.
flag: display test points	If checked, the test points of the current regulation will be shown in the UV Data View.
flag: road projection	Please see the next table and the corresponding picture below.

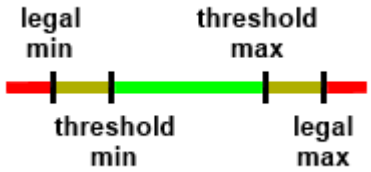


Figure 9.2: The LID Measure Dialog (II): Percentage Deviation Threshold

flag: percentage deviation	If checked, the text output is extended to show the attained percentage relative to the given legal min-/max-range. This extent can be an additional column (in most cases) or two additional lines when having a matrix measurement (e.g. for signal beams).
percentage deviation thresholds	It is possible to define some kind of tolerance using the percentage deviation, so the user is warned by a different color (dark yellow) if the measured values are *too close* to the legal limits. Using a threshold of (100,100) will set the threshold limits identical to the legal limits, so the tolerance interval is zero and the percentage deviation works as before. The entered values are the absolute deviation of the respective legal limits, this is almost identical to the calculation of the percentage deviation. A value of 110 for the threshold minimum means that the green interval starts at 110% of the legal minimum. A value of 85 for the threshold maximum means that the green interval ends at 85% of the legal maximum. The three colours work as follows:

- The measured values which lie outside the interval of the legal values (i.e. below the legal minimum or above the legal maximum) are displayed in red, because this measurement does not fulfil the regulation at all.
- The values in $[min_{legal}, min_{threshold}]$ or $[max_{threshold}, max_{legal}]$ fulfil the regulation, but the measured value is *too close* to the legal limit, they are displayed in yellow.
- The values in $[min_{threshold}, max_{threshold}]$ fulfil the regulation and they lie within the user-defined range of tolerance, they are displayed in green.

The colour of the worst single measurement is used for the colour of the summary, too.

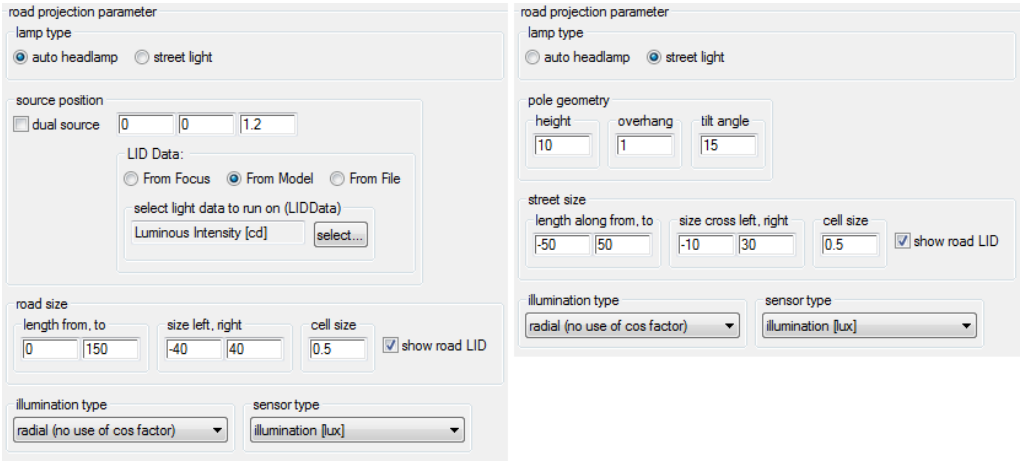


Figure 9.3: The LID Measure Dialog (III): Road Projection

radio button: lamp type	One can choose either *auto headlamp* or *street light*.
flag: dual source	This option determines whether there are one or two sources.
source 1/2 position	Sets the position of the light source in cartesian coordinates.
pole geometry	Determines the position of the lamp relative to the street via *height*, *overhang* and *tilt angle*.
LIDData	The data for the projection can be a UVView in focus, a LIDData from the TreeView or an external file.
road/street size	Determines the length, the width and the resolution of the road respective street section to be shown and calculated.
flag: show road LID	If checked the LID on the road is shown after applying the Test Table.
illumination type	One can choose *radial (no use of cosine factor)*, *vertical*, *half spherical*, *cylindrical* or *semi cylindrical*.
sensor type	Chooses either *illumination [lux]* or *luminance [cd/m²]*.

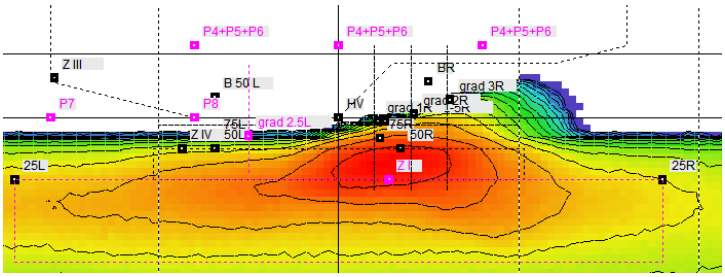
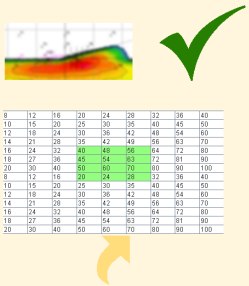


Figure 9.4: LID test points in a UV data view for a low beam application

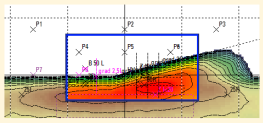
9.1.2. Apply Last Selected Test Table

This menu entry offers a convenient shortcut to apply the test table which has last been selected by Section 9.1, “LID Measuring”. The command works directly on the UV data view in focus, without any further dialog.



9.1.3. Measure Point or Zone

This menu entry offers a collection of zone specific measuring tools which may me applied on the UV data view in focus. Apart from the flux calculations (the results of which are in lumen), all types of measurements may be performed in *candela*, *lux on 10m* or *lux on 25m*.



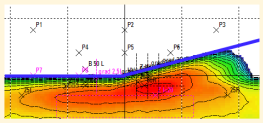
point measure	Measures the intensity at a specific point in the UV data view. Supply the point in the dialog.
zone measure	Measures the minimum and maximum intensity in a specified zone in the UVView. The positions corresponding to the min- and max-values are reported as well.
find min (max)	Finds the overall LID minimum (maximum) value. The uv-position corresponding to the min (max) value is reported as well. This dialog is non-editable and requires no input by the user.
zone mean measure	Measures the mean intensity in a specified zone in the UVView.
integral flux	Computes the overall LID flux in lumen. This dialog is non-editable and requires no input by the user.
flux in zone	Computes the LID flux in lumen in a specified rectangular zone.

9.2. LID Modeling

This section offers a few tools to model uv-data-views according to regulations.

9.2.1. Trim Cutoff Lines

This menu entry pops up a dialog with several buttons, each one offering a specific LID cut-off (like ECE cut-off for low a beam application etc.) for the UV data view in focus. This does not change the geometry, it is only applied onto the LID.



Zone III ES CutOff(0L)
Zone III LES CutOff(0R)
Zone III ES CutOff(-0.57L)
Zone III LES CutOff(-0.57R)
Zone III 45 ES CutOff(0L)
Zone III 45 LES CutOff(0R)
Zone III 45 ES CutOff(-0.57L)
Zone III 45 LES CutOff(-0.57R)
Zone III R98 ES CutOff(0L,1R)
Zone III R98 LES CutOff(0R,1L)
Zone III R98 ES CutOff(-0.57L,1R)
Zone III R98 LES CutOff(-0.57R,1L)
horizontal CutOff(0.0)
horizontal CutOff(-0.4)
horizontal CutOff(-0.57)
FMVSS108 VOL CutOff(0.0)
FMVSS108 VOL CutOff(-0.4)
Zone IIIa R123 Class C or V ES
Zone IIIa R123 Class C or V LES
Zone IIIb R123 Class W or E ES
Zone IIIb R123 Class W or E LES

Figure 9.5: The Trim LID Dialog

The following picture shows an example of a non-automotive LID before (left part) and after (right part) the cutoff "Zone III ES Cut-Off (-0.57L)" has been applied.

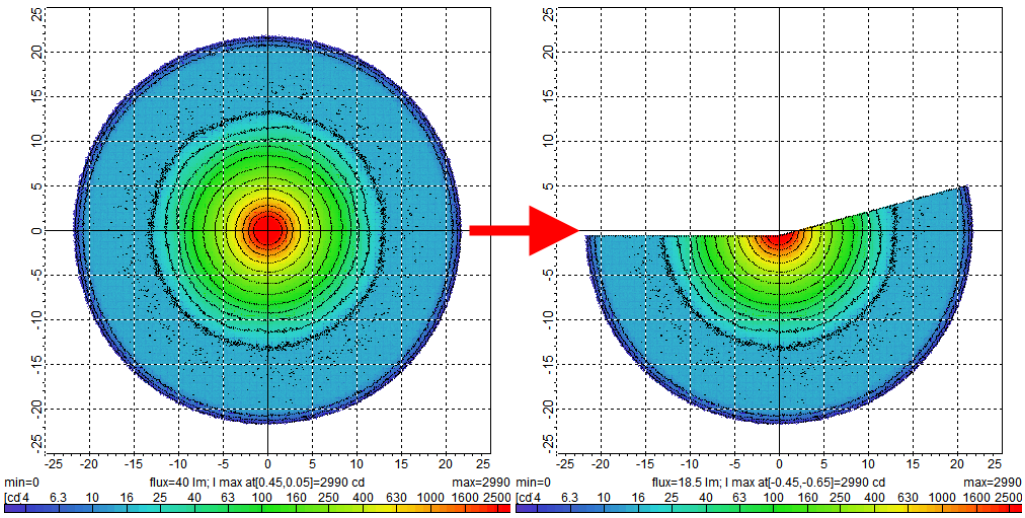
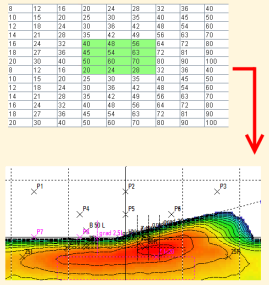


Figure 9.6: A LID before (left) and after (right) applying a trim.

9.2.2. Interpolate LIDs from Test Tables

This menu entry pops up the test table dialog which is described in more detail in Section 9.1.1, “Apply Test Tables”. In contrast to evaluating existing LIDs, however, it uses interpolation methods to compute a new idealized model LID which matches the selected regulation in the test table.



The picture right below shows two examples created with this tool:

- Upper part: ECE → front turn signal lamps ECE → R6 Cat. 1 left
- lower part: USA → Daytime running light → SAE J 2087 tab. 2 +zones

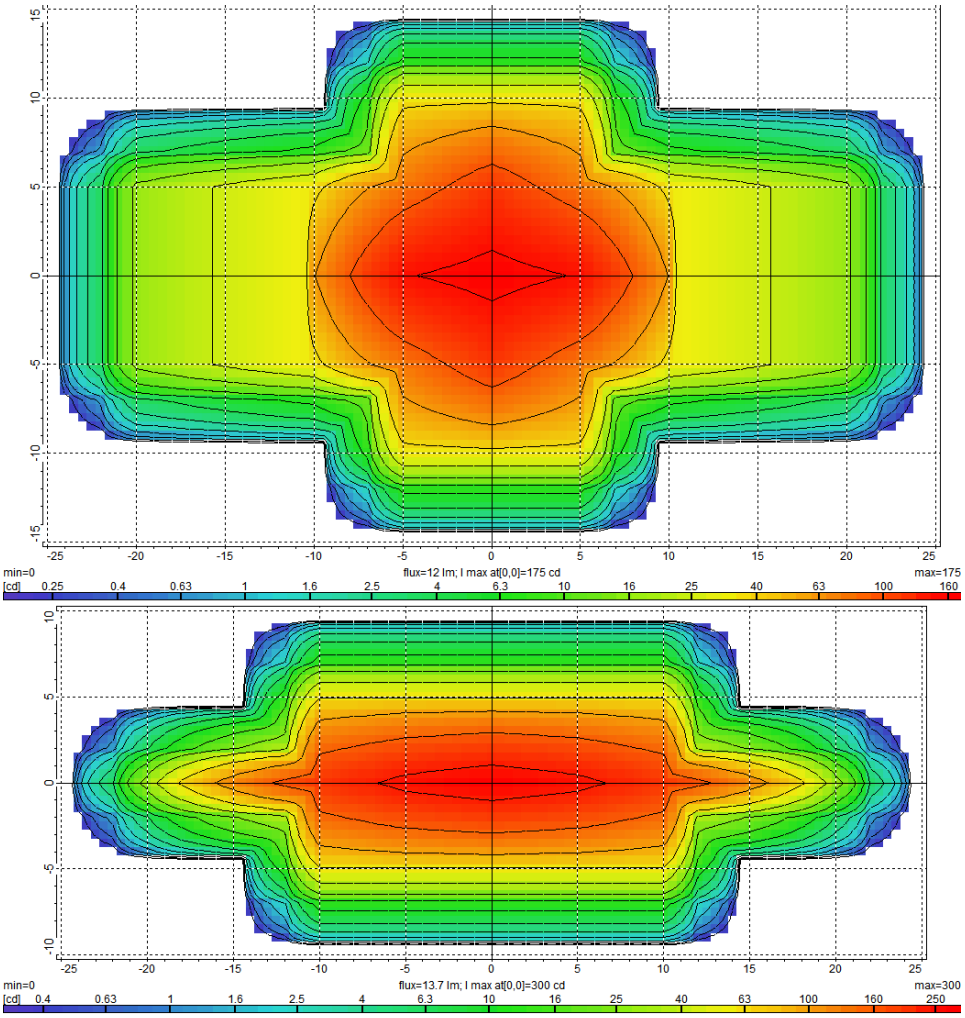
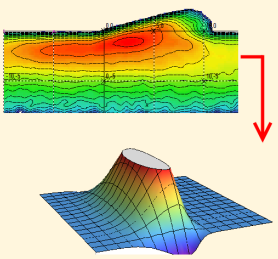


Figure 9.7: Two interpolated LIDs created with the tool.

9.3. LID Data Conversion

9.3.1. Surface from LID Data

A tool to compute a surface from the uv-data-view in focus. The surface is created (in a special geometry view) as a piecewise linear NURBS surface which may be used elsewhere in LucidStudio.



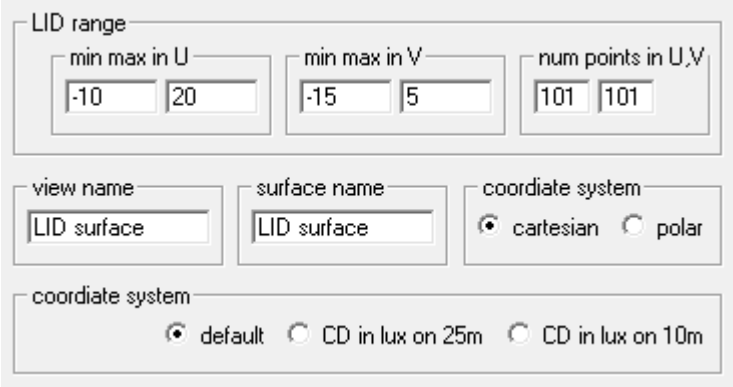


Figure 9.8: The Create Surface from LID Dialog

min/max in u, v	Determines the ranges of u and v respectively.
number points in U, V	The number of points to build the curve.
view name, surface name	Determines the names for the view and the surface respectively.
radio button: coordinate system	Determines whether the curve shall be created in cartesian or polar coordinates.
radio button: coordinate system	Chooses the range where the illuminance is to be measured.
flag: colored display	Determines whether the figure is shown in color or as a wireframe.

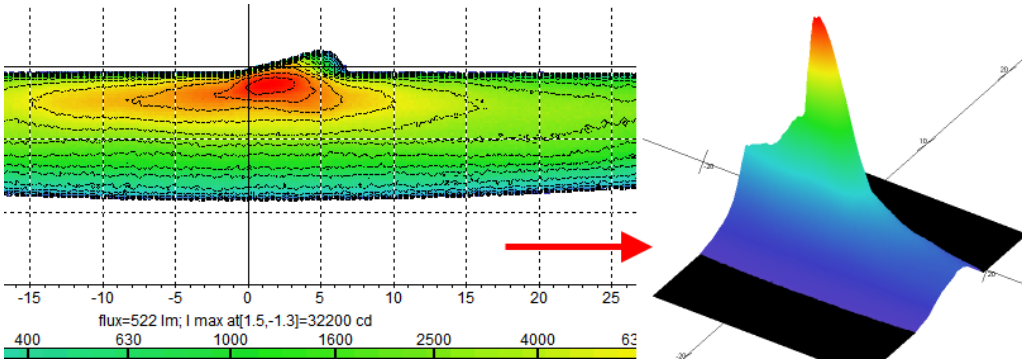
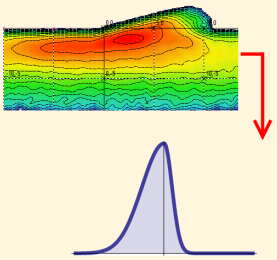


Figure 9.9: Example of a surface from an ECE low-beam LID

9.3.2. Section Curve through LID Data

A tool to compute a horizontal or vertical section curve from the UVView in focus. The curve is created (in a special geometry view) as a NURBS curve which may be used elsewhere in LucidStudio.



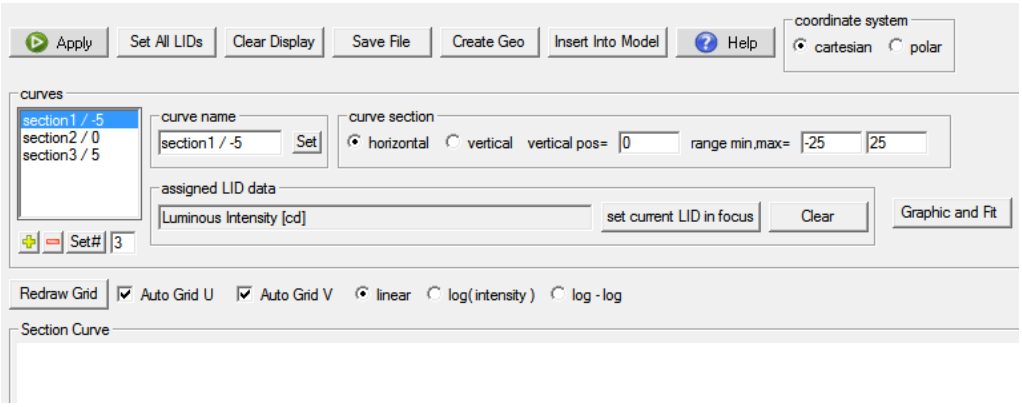


Figure 9.10: The Create Section Curve from LID dialog

button: apply	If pressed, the made changes are adopted and the section curves are (re-)drawn. The altered curve is not simply added to the drawing area, but replaced.
button: set all LIDs	Mark a LID and press this button to quickly change the used LID for all section curves in the list.
button: clear	If pressed the drawing area is cleared.
button: save file	If pressed the data in the drawing area is saved into a text file.
button: create Geo	If pressed the section curves are displayed in a GeoView the same way as the old dialog did.
button: insert into model	If pressed the dialog with all its settings is inserted into the model as a design feature.
radio button: coordinate system	Determines whether the curve shall be created in cartesian or in polar coordinates.
curve	Here all section curves are listed. A single section curve can be added via pressing '+', while '-' will delete the currently marked one. With *set#* the total number of curves can be set.
curve name	Here the single section curves can be renamed, changes are applied by pressing *set*.
radio button: curve section	Determines whether the section curve shall be calculated in horizontal or vertical direction. The position and the range are determined by the horizontal/vertical position and the min/max values.

assigned LID data	Shows the used LID for this section curve. It can be changed by marking a LID and pressing *set current LID in focus*.
button: graphic and fit	If pressed a small dialog opens. Here some additional drawing parameters like e.g. the color are adjustable. Apply this by pressing *OK*.
button: redraw grid	After having altered the drawing area (e.g. move or zoom), it can be redrawn using the current settings.
flag: auto grid u/v	The values to be used for the grid can be taken automatically from the LID or entered manually.
radio button: scaling	This radio button determines the scaling of the grid. The default setting is *linear*, here the grid is drawn in the usual linear style. The choice *log(intensity)* keeps the u-axis linear, but changes the v-axis (i.e. the intensity) to logarithmic. The third choice *log-log* changes both axis to logarithmic.

The following picture shows an example with the original LID from an ECE low-beam (upper part), a vertical cut curve taken at u=-5 (blue curve) and a horizontal cut curve taken at v=-2 (red). The two section curves are marked in the original LID.

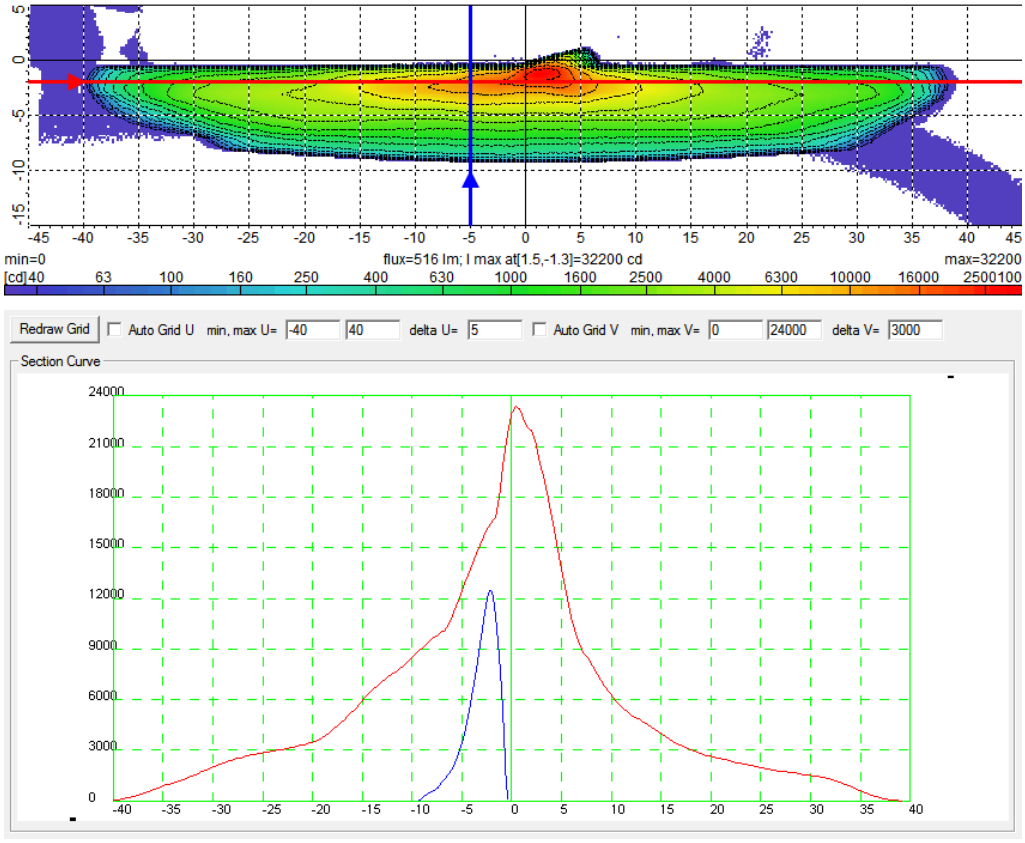


Figure 9.11: Example of two section curves taken from a low-beam ECE LID

9.4. Analysis of 3D Geometry

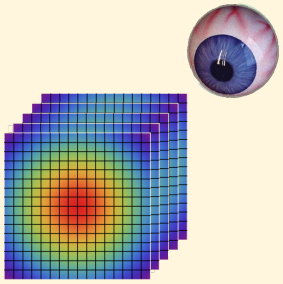
9.4.1. Bounding Box

The bounding box does not have a dialog or any options or parameters. It can be applied onto the currently active GeoView and considers all extended 2D- and 3D-objects like curves, surfaces and point clouds, but no single points or vectors.

The bounding box is the smallest possible cuboid containing the entire model, the single planes perpendicular to the global axis system. It states the minimum and maximum values for all three dimensions respectively, together with their respective extent.

9.4.2. Visualize Volume Sensor

Uses planar sections to visualize volume sensor light data in the geometry view. The light data must have been collected in a previous simulation run.



feature name	The name of the feature, this is the identifier in the TreeView
pull-down menu	In case there is more than one sensor in the model, it can be selected from the list.
min value, max value	Defines the range of the light data to be displayed within the geometry view. A value of '-1' means unlimited, all values in the GeoView below the minimum are shown in black, all values above the maximum are shown in red.
plane orthogonal to	The planar sections on which the light data will be displayed may be positioned orthogonal to either the x-, y-, or z-axis.
show plane #	Switches to the plane with the assigned index. You can use the arrow keys to scroll in the planar sections sets. The position of the plane inside the volume sensor is displayed below as info text.
button: show plane	Shows the currently selected planar section in the volume sensor display.
button: add plane	Adds a planar section to the volume sensor display.
button: show max planes	Determines the three planar sections with the respective highest intensity values and shows them in the GeoView.
button: start animation	If pressed, the currently active planar section is moved forwards and backwards in the range of the border defined below.
button: stop animation	If, pressed, the animation is stopped and the last position is kept.
button: clear planes	Removes all planar sections from the volume sensor display.

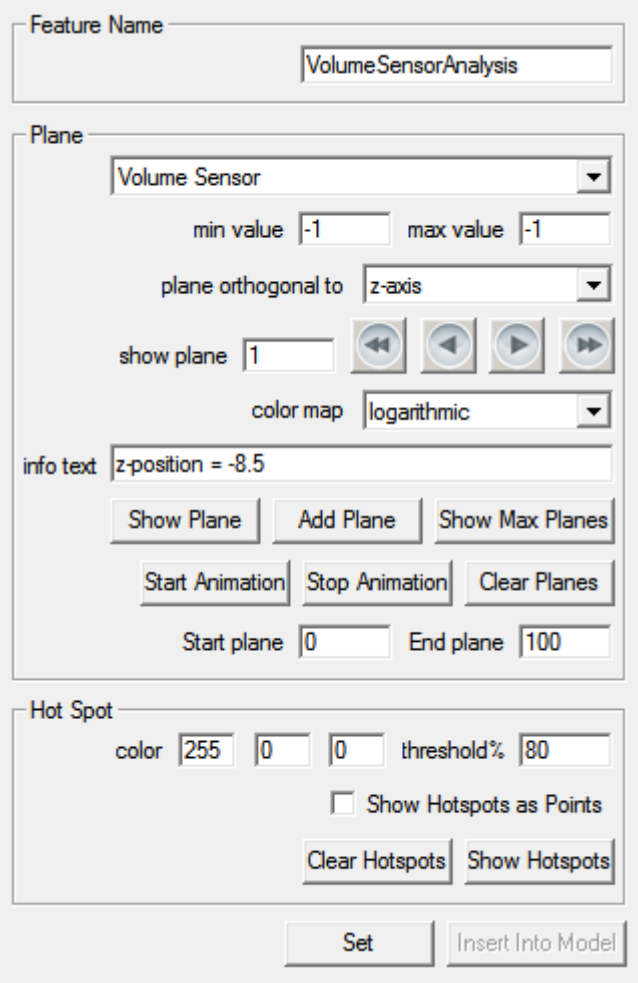


Figure 9.12: The Visualize Volume Sensor Dialog

start/end plane	Determines the border of the animation. The range is not taken from the sensor range, for all three directions the range is [0,100].
hotspot color	RGB values determining the display of the hot spots in the geometry view.
hotspot threshold	The percentage of the hotspot threshold. Only regions having intensity values above this threshold are defined and displayed as hot spots.
flag: show hotspots as points	If checked, the hotspots are not shown as cubes, but points.
button: clear hotspots	Removes the display of hotspots.
button: show hotspots	Displays the hotspot locations (i.e. locations with intensities above the hotspot threshold value).
button: set	If pressed the dialog with its current values and parameters is applied.
button: insert into model	Adds this analyse into the model, so the dialog can be opened again from the TreeView.

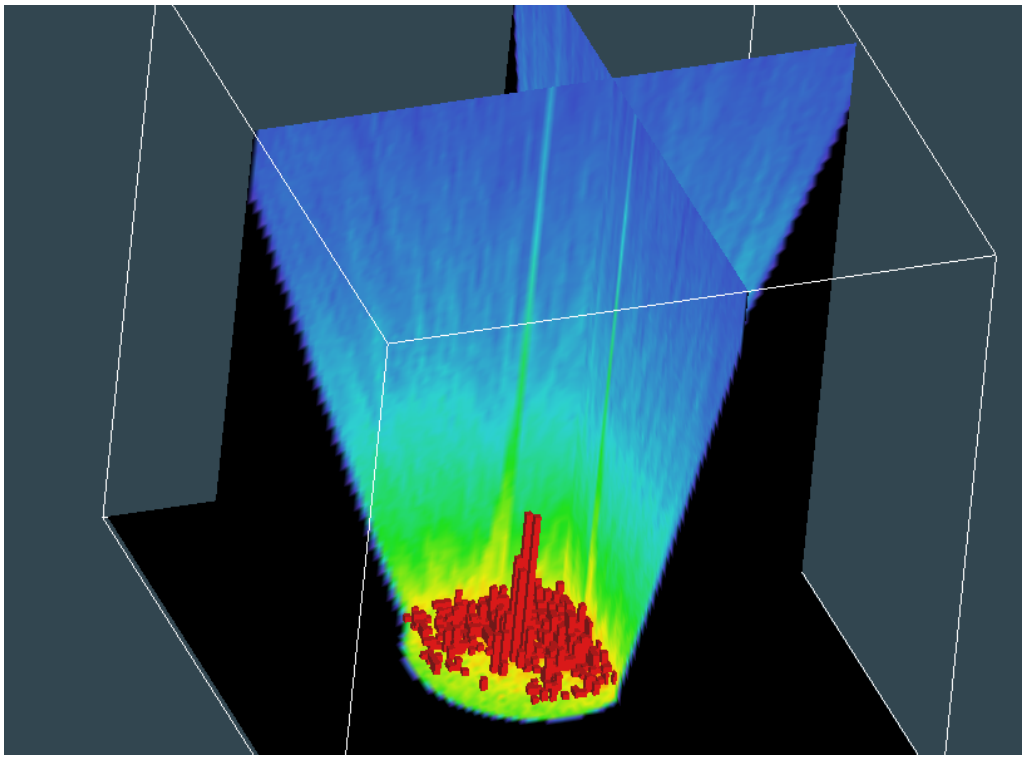
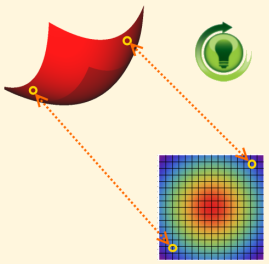


Figure 9.13: Example: The volume sensor including its three max planes plus hot spots

9.4.3. Restore Rays from Ray History Sensor

This dialog revives rays from a previously filled ray history sensor: The user may specify a region in the UVData view in focus, as a result all (or some) rays that reached that region during a previously performed Monte Carlo simulation will be displayed in the geometry view. See Section 7.4.3, “Ray History Sensor” for a description of how to create and handle ray history sensors.



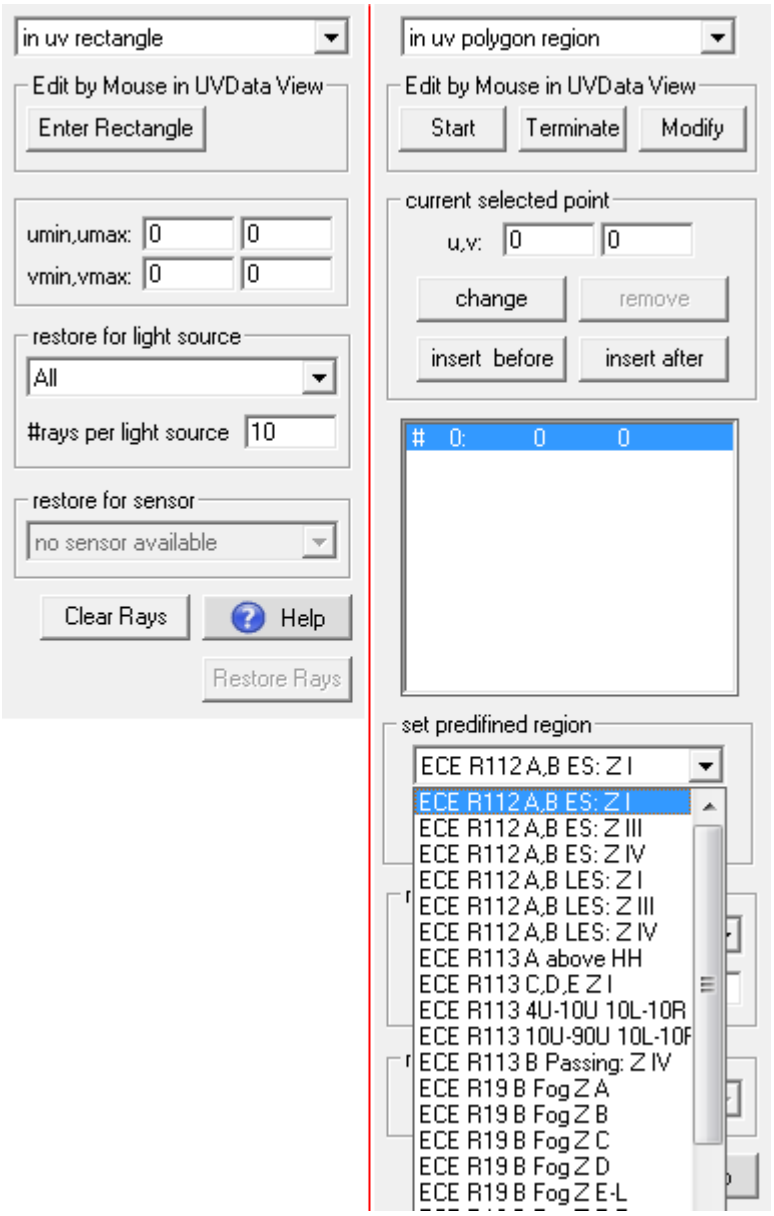


Figure 9.14: The Restore Rays from Ray History Sensor Dialog

	range selector	Specify here the type of uv-data range for the ray reconstruction. According to the selection, range parameters may be entered. Available range types are • *at uv*: A single uv-point, only rays that hit the sensor at this point (with a small radius) will be displayed. • *in uv rectangle*: A rectangular uv-area. Rays that hit this area will be displayed. • *in uv polygon region*: A polygonal uv-area: Rays that hit this area will be displayed. The polygon may be either entered via its nodes (which will be displayed in a list then), or regions specific to automotive regulations (like ECE R19 FOG Z A for a fog lamp) may be selected directly from a pull-down menu.

	• *at index*: Each ray in the ray history sensor may be selected directly by its index (starting from 0). Please keep in mind that only those rays can be restored which have been recorded by the sensor, i.e. which are within the sensor area.
restore for light source	A selector which may be used to restrict the ray reconstruction to specific light sources. The default setting is *All* to get rays from all light sources at a time.
#rays per light source	Restrict the number of reconstructed rays for each light source. Adjust the number to a compromise of enough rays and clarity.
restore for sensor	If you use more than one ray history sensor at a time, select here the sensor from which you want to reconstruct rays.
button: clear rays	Clears all reconstructed rays from the current model's 3D geometry view. (This is automatically done when pressing "restore rays".)
button: restore rays	Reconstructs the rays from the selected ray history sensor and displays them in the current model's 3D geometry view.

The picture right below shows an example with a low beam pattern used. The reflector to create the pattern was taken from the model gallery without changes ("08)Low Beam-ECE R112 oval H11.lug" or #1-1-1-8). The sensor ranges are u=[-10,10] and v=[-10,5] with a resolution of $\Delta_u=\Delta_v=0.1$ and type A.

The parameters used to restore the rays are: Rectangle range with $u_{min}=0$, $u_{max}=10$, $v_{min}=0$, $v_{max}=2$ and 20 rays per light source. The region is marked in orange in the LIDView. After pressing "restore rays" 20 rays are displayed in the GeoView (the intersection area on the reflector is marked in orange too).

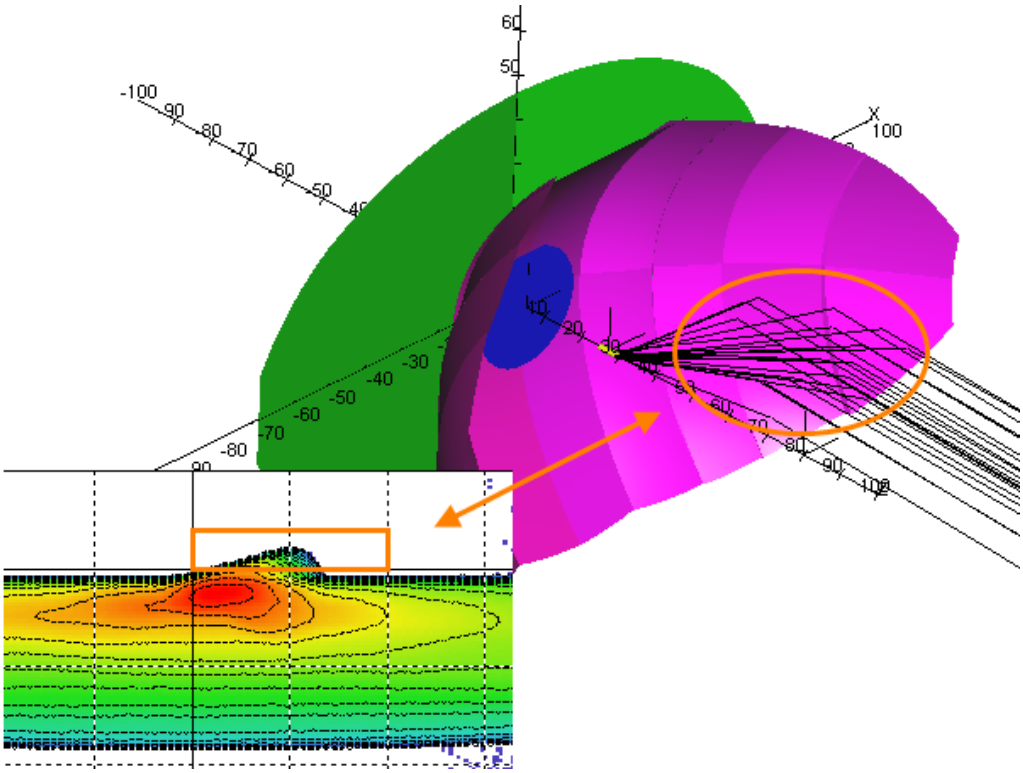
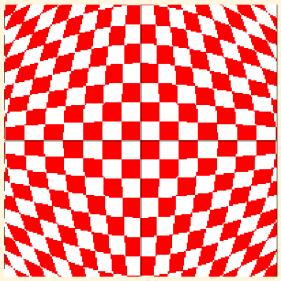


Figure 9.15: Typical Result of a Ray History Sensor Query

9.4.4. Ray Deviations and Checkerboard Images



Creates a ray deviation using a checkerboard image to visualize the deterioration behaviour of a lens. This can be used e.g. to evaluate the behaviour of a cover disc (which in ideal case should not deflect the light at all).

A ray deviation diagram is a two dimensional data image that displays the deflection of a two dimensional parallel light bundle by the lens. The image shows the simulation result of an observer's view (from a certain distance) on a checkerboard through the lens.

There are some examples provided with LucidShape, especially one with 'good' and one with 'bad' shapes (goodShapes.lug / badShape.lug) to illustrate the very different behaviour of a bad lens compared with a corrected one. These .lug files and some further .lid files can be found in the installation folder at ..\LucidShape\samples\rayDeviationByLens\.

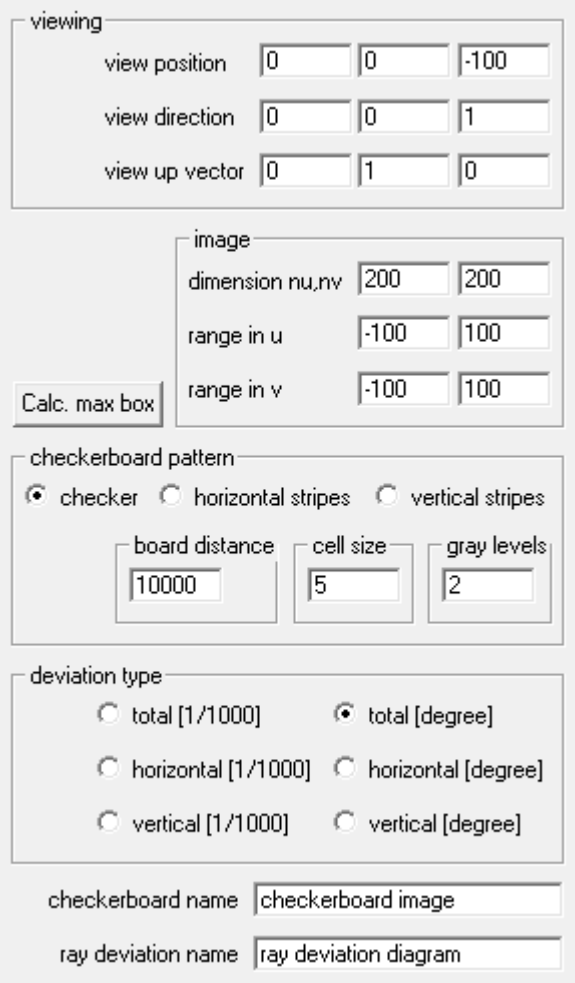


Figure 9.16: The Ray Deviation and Checkerboard Dialog

viewing	These parameters determine the position of the viewer, the view and the up direction of the geometry.
dimension n_u, n_v	Determines the number of cells in each direction, i.e. the resolution of the checkerboard-grid.
range in u,v	Determines the extent of the checkerboard image and the ray deviation diagram in u- and v-direction.
button: calculate max box	If pressed a surrounding box with corresponding values for the resolution is calculated.
radio button: checkerboard pattern	Sets the type of the pattern to either checker, horizontal or vertical stripes.
board distance	The distance between the checkerboard and the lens to be examined. This value may have a positive or negative sign, so the checkerboard image may be positioned at both sides of the lens. Of course, to see any resulting deterioration in the pattern, the lens must be positioned between the checkerboard and the observer.
cell size	The width of the squares (resp. stripes) in the checkerboard.

gray levels	The number of gray levels in the checkerboard. Increasing the number allows to see more details of the deviation.
deviation type	Defines the type of deviation in the ray deviation diagram. You may select between deviation in degrees (with respect to the incoming parallel ray bundle) or total deviation in mm on the viewing position plane. In addition, horizontal or vertical deviation may be selected for both modes. The results for "total [1/1000]" and "total [°]" are identical with exception of the map scale of course.
checkerboard name	The name of the checkerboard image.
ray deviation name	The name of the ray deviation image.

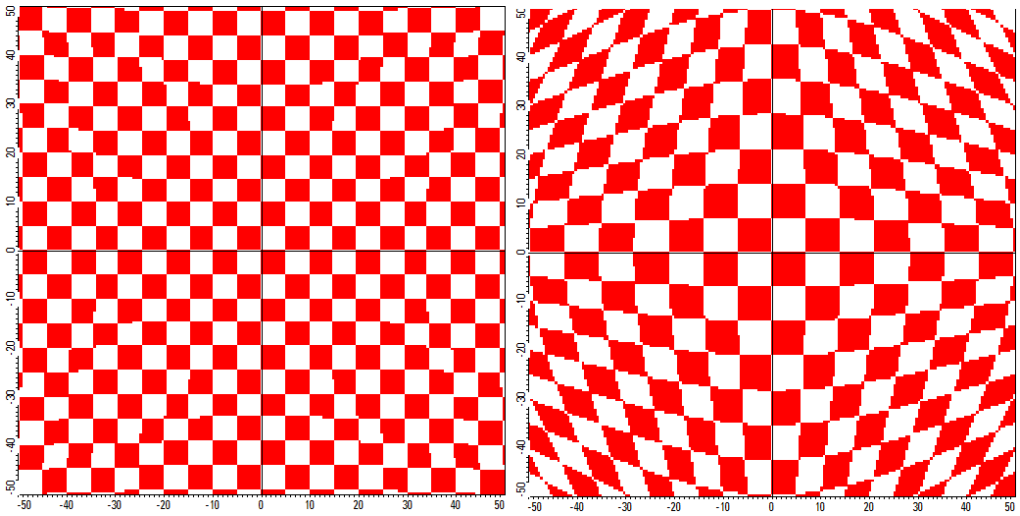
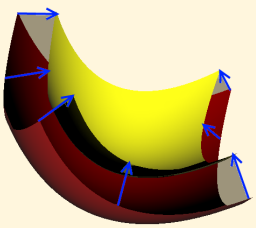


Figure 9.17: Example: Checkerboard undistorted (left) and distorted by lens (right)

9.4.5. Wall Thickness Diagram



For a given start surface, take the surface's normal at a (u,v) parameter value and calculates if the corresponding normal ray (i.e. the ray starting at this point on the start surface with the normal as direction vector) intersects another target surface. If there is an intersection, the distance or wall thickness between the start and the target surface at (u,v) may be defined as the computed intersection distance.

This tool computes a distance diagram between two given surfaces and displays the diagram as a uv-data view. It should be clear that this calculation only makes sense in cases where such intersections exist over a substantial portion of the start surface's uv-range.

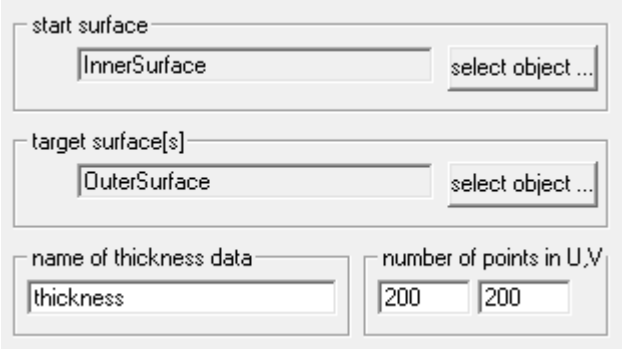


Figure 9.18: The Wall Thickness Diagram dialog

start surface	The start surface for the thickness calculation.
target surface	The target surface for the thickness calculation.
name of the thickness data	Caption used for the generated UV data view which contains the thickness diagram.
number of points in u,v	Number of points to uniformly sample the uv parameter range of the start surface. The distance between the start and target surface is computed at these grid points.

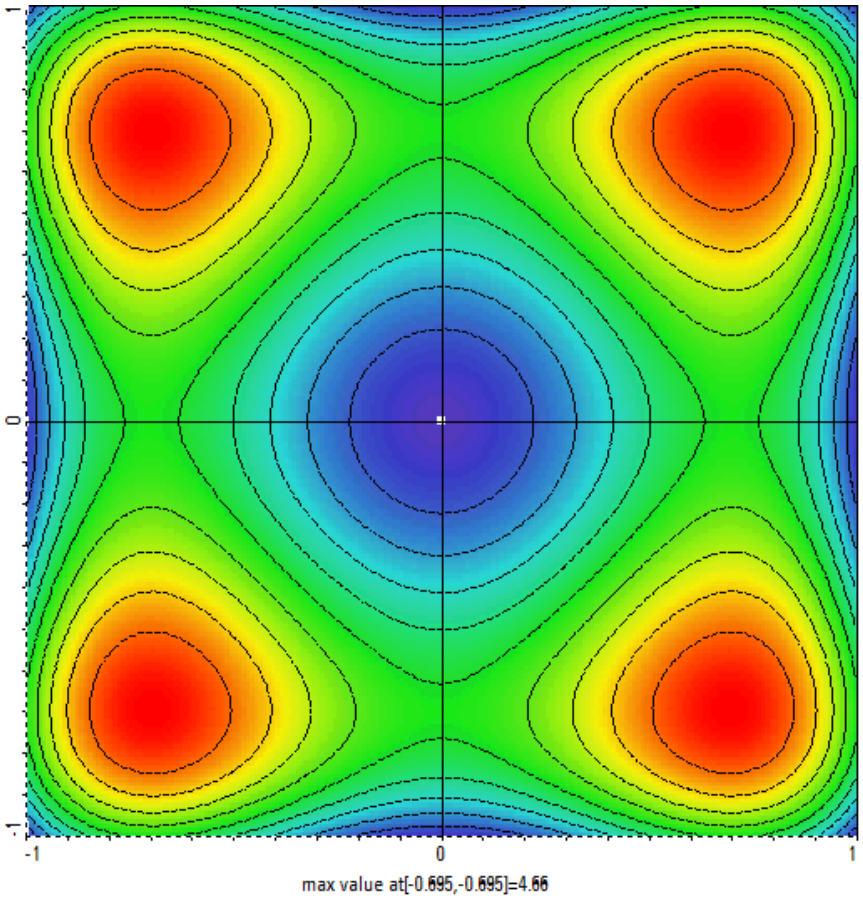


Figure 9.19: Example: Distance between the surfaces ($x=5ou$, $y=5ov$, $z=u^2+v^2$) and ($x=5ou$, $y=5ov$, $z=u^4+v^4$) in the interval [-1,1] for both u and v.

9.5. Benchmark test tool for TC4-45 or self defined assessments

This tool has been implemented for assessing the photometric performance of vehicle headlights such as road scene illumination and glare limitation as well as lane guidance and the detection of pedestrians and objects.

A special TC4-45 assessment is available under "Menu → Analysis → Open Benchmark" (..\LucidShape\classes\benchmark\prototype_tc4-45_vo.lub), open Benchmark allows to open self-defined ones.

The content of the TC4-45 is not described here, it is supposed to be known to the user when checking LIDs according to this assessment.

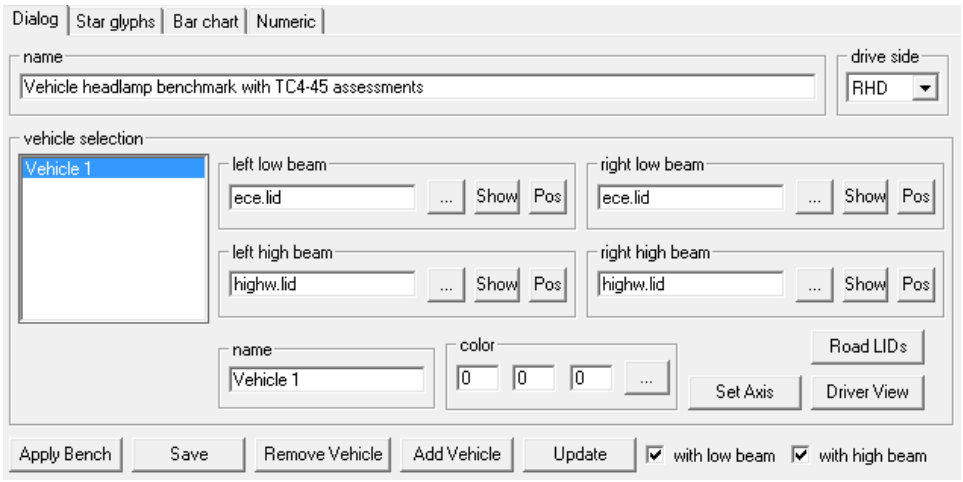


Figure 9.20: The main Benchmark Dialog

With this dialog, it is possible to select sample vehicles from a list box, assign LIDs for left/right high/low beam to them, choose a special position for each lamp and perform an assessment check pressing the "Apply Bench"-button. The result of the benchmark test can be examined in three different representations: Star glyphs, Bar Chart and a numeric table. Furthermore it is possible to view the corresponding road LIDs and driver view. The latter one opens a new dialog which allows to select special assessments which are visualized in the driver view representation.

name	The name of the benchmark to be used. The only one which is implemented so far is TC4-45, and only for low beam LIDs.
drive side	The assessments can be applied for both LHD (=left hand driving) and RHD (=right hand driving) traffic.
vehicle selection	Allows to switch between the different vehicles.
left/right high/low beam	Pressing the "..." buttons opens a file selection box in order to select the LID files to be checked. "Show" displays the LIDs in LucidShape. Left and right refers to the left and right headlamp of the car. It has nothing to do with left/right hand traffic!
button: pos	If pressed a dialog for the mounting properties pops up, here the position of the lamps can be altered. They are measured relatively to the center of the car.
name	The name of the current sample vehicle.

color	The color display in Star Glyph and Bar Chart representation.
button: set axis	The axis system for the vehicle position, the positions for the LIDs are relative to this position. (For a detailed description please refer to Section 4.10.1, “Create Axis System ”.)
button: road LIDs	If pressed the above specified LIDs are used to create four bird's eye views and a view on a wall. The therefor used parameters are shown in the respective title bar of the views.
button: driver view	If pressed two windows are created, a dialog for making an assessments selection pops up (see the picture right below).
button: apply bench	Performs the benchmark test.
button: save	Saves the current settings using the entered benchmark name in the upper dialog part.
button: remove vehicle	Removes the marked vehicle from the selection list.
button: add vehicle	Adds a further vehicle to the selection list.
button: update	Accepts current changes made in the dialog box.
flag: with low beam	If checked, the low beam assessment is applied.
flag: with high beam	If checked, the high beam assessment is applied. Until now this benchmark is under progress and has not been passed as law (12/2011).

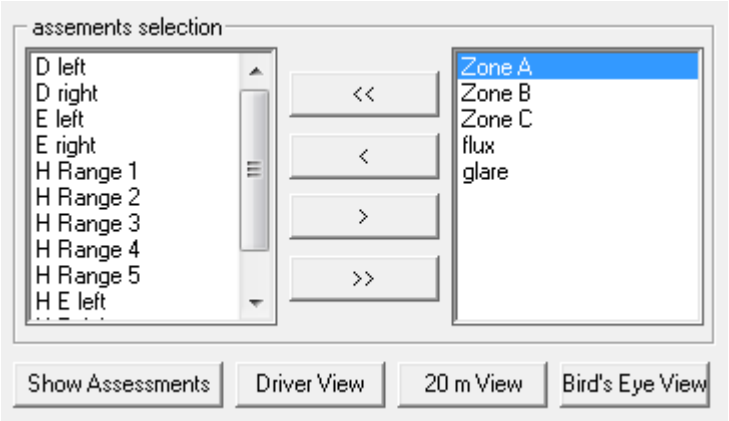


Figure 9.21: Dialog box for Assessment Display and Views Selection

Driver View shows the light intensity distribution on the road. With the dialog box below it is possible, to show different assessments in the street scene. Here the assessments Zone A, Zone B, Zone C, Flux and Glare are selected. The single items can be shifted from the left list to the right and vice versa using the arrow-buttons. It is also possible to show a 20m view and a bird's eye view.

The following two pictures show the three graphical situations on the road (upper) and the three different analysis tools (lower picture). All assessments are done only for the low beams.

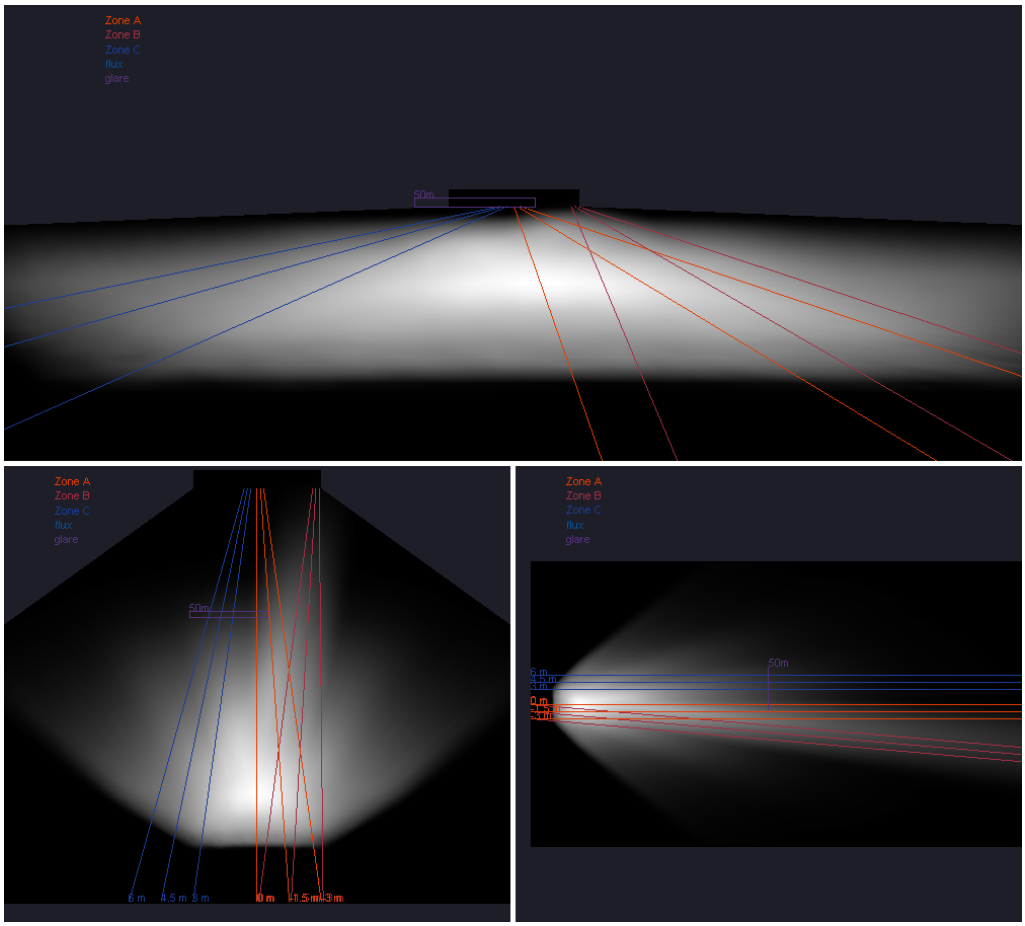
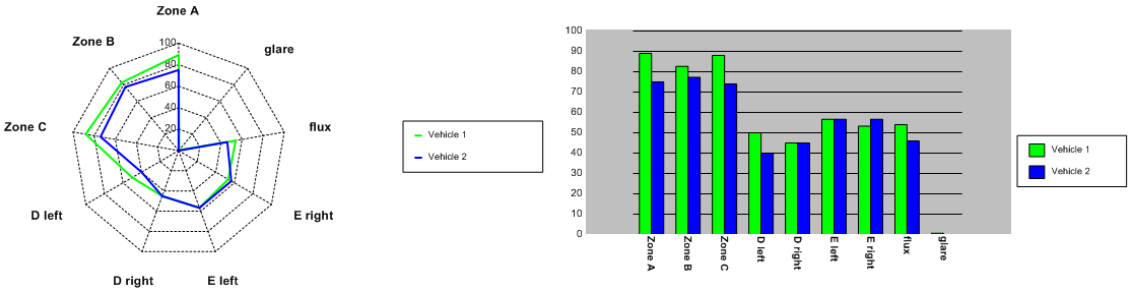


Figure 9.22: Road simulation: The upper part shows the Driver View, lower left the 20m View and lower right the Bird's Eye View, all with the above chosen assessments.



Low Beam Absolute Rating

test criteria	Zone A	Zone B	Zone C	D left	D right	E left	E right	flux	glare
Vehicle 1	89	124	44	10.0	9.0	8.5	8.0	1080	0.01
Vehicle 2	75	116	37	8.0	9.0	8.5	8.5	920	0.00

Figure 9.23: Road simulation: The upper left part shows the Star Glyph diagram, the upper right the Bar Chart and the lower part the Numeric output.

9.6. Bitmap Tools

9.6.1. Bitmap Show

This tool displays bmp pictures from a given folder, either statically or as a movie where the display speed can be defined via a timer interval. May be used to display the bitmaps generated in Section 9.6.2, “3D Snapshot”, for example.

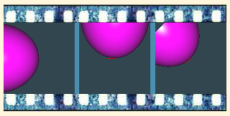
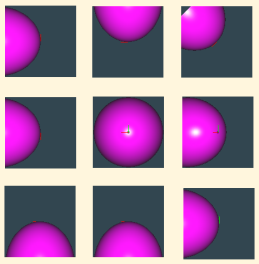


Figure 9.24: The Bitmap Show Dialog

button: setting	Select here one image in the wanted path to be displayed (*.png, *.bmp or *.jpg). The selected image also serves as a starting object for a movie animation.
|<-	Jumps to the beginning of the folder.
<,>	Jumps one frame backward respective forward.
P, S	Plays respective stops the movie.
default size, user size or scale	Select between the default size (original picture size), user size (applies the specified values) or scale (applies the scale factor onto the original size).
timer interval	Specifies the delay between two sequent pictures in milliseconds.

9.6.2. 3D Snapshot

This tool generates a two-dimensional series of screen shots from a model displayed in the geometry view. The views are defined by a resolution, a horizontal and a vertical angle range. The single views are saved in the specified folder in the .png file format. They can be used in a subsequent step to be displayed as a movie (see the subsection right above).



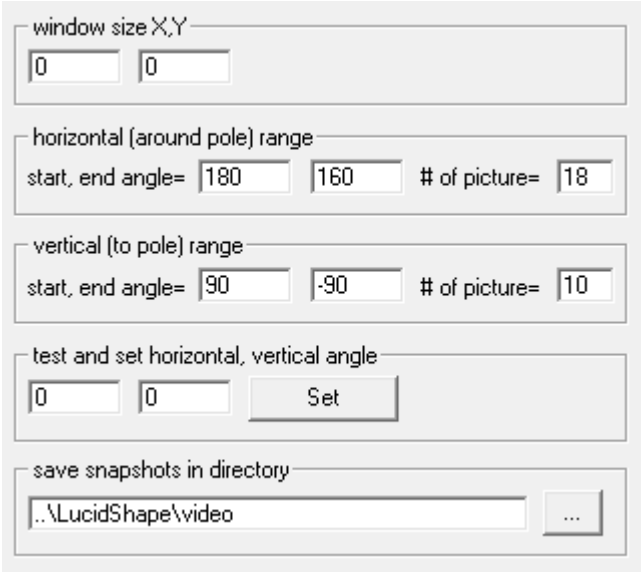


Figure 9.25: The 3D Snapshot Dialog

window size x,y	Defines the (x,y)-extent of the window the snapshots are taken from. If a window is set into focus, whose values will be adopted automatically (these values will be updated every time the windows is highlighted).
horizontal /vertical angle range	The respective starting and ending angle values of the series to be taken.
# of picture	The number of pictures to be made in each angular direction. The default settings 18 and 10 will produce 180 pictures in total.
test and set horizontal, vertical angle	Sets the model in the currently focussed geometry view to the designed horizontal and vertical angle.
save snapshots in directory	Enter here the directory the pictures shall be saved into.

Chapter 10. LIDEditor

10.1. Overview

This chapter handles the LID Editor and the LID Editor Gallery. The editor is for editing and composing LID files (light intensity distribution), the gallery is kind of an example library which contains several pre-defined setups.

The LID Editor is a tool for the theoretical examination of the question: "How can I achieve a beam pattern with a given set of modules?" To be able to solve this question, the editor can add and compose different LID files. After the files have been loaded, the individual LIDs can be customised (stretched, shifted, rotated, scaled and mirrored).

Assume, one has to (or like to) invent a perfect theoretical light intensity distribution (LID) of some kind. On the other hand, one has different modules, each one providing different LIDs. One can ask the question, can I combine the individual light intensity distributions from the modules in order to get the perfect desired LID? And, if so, which mathematically operations on each LID have to be performed for this goal?

Instead of running a simulation with a large number of modules, one can simulate the individual modules and add their results with the LID Editor. But the combination of individual LIDs should not replace the simulation of the model as a whole. Using the LID editor can be the first step before the development work begins.

10.2. LID Editor Gallery

The gallery can be launched via "Menu → (UV)Analysis → LID Editor Gallery" and contains several examples, i.e. pre-defined setups. The dialog is split into two selection lists. The upper part is the main intended purpose of the beam or light, in the lower part the detailed specification can be selected. Pressing *load* will load the currently selected entry in a new model, while *add* will add this selection into the currently active model.

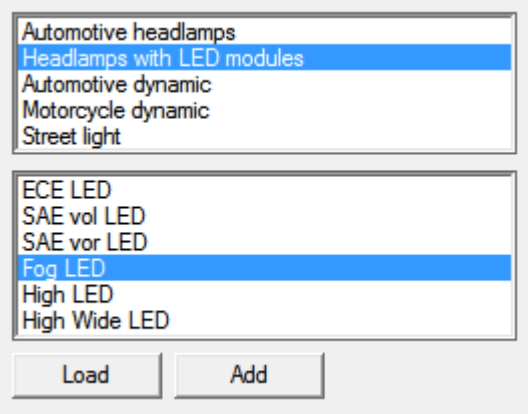


Figure 10.1: The LID Editor Gallery

10.3. LID Editor

The editor can be launched via "Menu → (UV)Analysis → LID Editor", here no pre-defined LIDs are included. In case the dialog will not pop-up automatically, please open it from the entry in the TreeView.

10.3.1. Loading LIDs

In this tab it is possible to set the number of lid files to be used and to enter the names of them. The short names appear in the next tab.

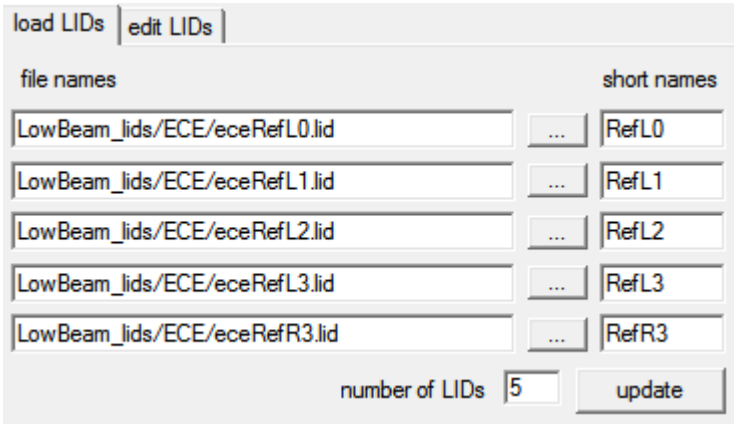


Figure 10.2: The LID Editor: Load LIDs

file names	The name and the path of the LID file which is used. When pressing the '...'-button, the usual file selection box opens.
short names	The short name of the LID file in use. They appear in the edit section as the identifier in 'use LID'.
number of LIDs	The number of LID files which will be used, it corresponds to the number of entry lines in the dialog.
button: update	Applies the changes onto the dialog box, i.e. the number of lines.

10.3.2. Options and Parameters

The options in this dialog part determine the availability of the single parameters in the tab *edit LIDs* and whether the tab *dynamic sets* is available or not.

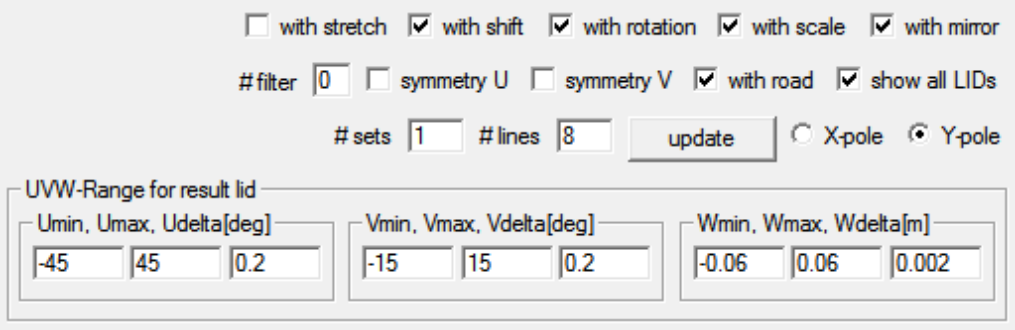


Figure 10.3: The LID Editor: Options and Parameters

flag: with stretch	If checked, the "stretch in -u,+u,-v,+v" dialog box is enabled.
flag: with shift	If checked, the "shift in u,v" dialog box is enabled.
flag: with rotation	If checked, the "rotate z" dialog box is enabled.
flag: with scale	If checked, the "scale" dialog box is enabled.
flag: with mirror	If checked, the "with mirror" dialog box is enabled.
# filter	Applies the specified number of filterings onto the resulting LID.
flag: symmetry in u	If checked the resulting LID will be mirrored at the u-axis, more precise it will be copied in the opposite direction (leading to a doubling of the flux).
flag: symmetry in v	If checked the resulting LID will be mirrored at the v-axis, more precise it will be copied in the opposite direction (leading to a doubling of the flux).
flag: with road	If checked, additional parameters are enabled to specify a bird's eye view, see description below.
flag: show all LIDs	If checked, applying will show the resulting LID and in addition all source LIDs. Multiple uses of one a single LID (e.g. mirroring) will not be shown.
number of sets	If set to 0, the LID editor is as described here, if set to a number larger than 0, an additional tab appears: Dynamic Sets (see description below).
lines	This value determines the number of lines (i.e. single LIDs) in the 'load LID' section.
button: Update	After changing '# sets' or '# lines', this button has to be pressed to apply the changes onto the dialog.
radio button: x-pole, y-pole	Displays the resulting LID either in the x-pole or y-pole system, please also refer to the chapter 1 in LucidTechnical.pdf (*Polar Angle Coordinate Systems*).
u_{min}, u_{max}, u_{Δ} [°]	Determines the range and the angular resolution of the u-range.
v_{min}, v_{max}, v_{Δ} [°]	Determines the range and the angular resolution of the v-range.
w_{min}, w_{max}, w_{Δ} [m]	Determines the range and the resolution of the dynamic set range.

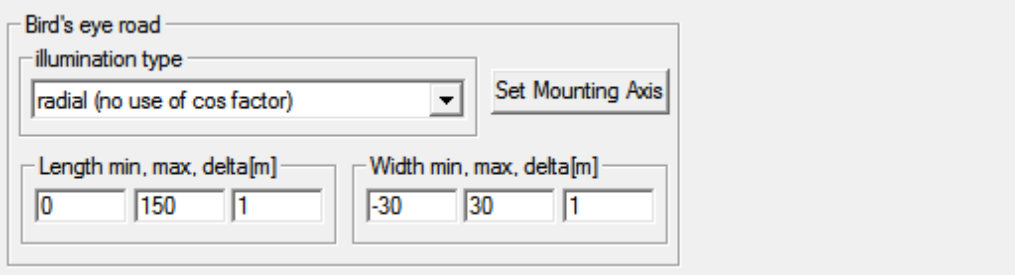


Figure 10.4: The LID Editor: Options and Parameters Bird's Eye View

pull-down menu: illumination type	Determines the type of the illumination used for the calculation, please also refer to the chapter 3.3 in LucidTechnical.pdf (*Types of Illuminance in LucidShape*).
button: set mounting axis	Opens a small separate dialog which allows to define the mounting coordinate system (position, orientation, etc.). The two buttons 'default street light' and 'default automotive' directly set the values to the respective standard values.
length min, max, Δ [m]	The min/max range of the street and the resolution in u of the resulting LID on the street in [m].
width min, max, Δ [m]	The min/max width of the street and the resolution in v of the resulting LID on the street in [m].

10.3.3. Editing LIDs

In this dialog part the previously loaded LIDs can be customised, the can be stretched, shifted, rotated, scaled and mirrored. With the settings shown in the picture right below, five different LIDs have been loaded (RefLo - RefR3). These five LIDs are used to create eight single LID-parts, the resulting LID is the sum of these LID-parts.

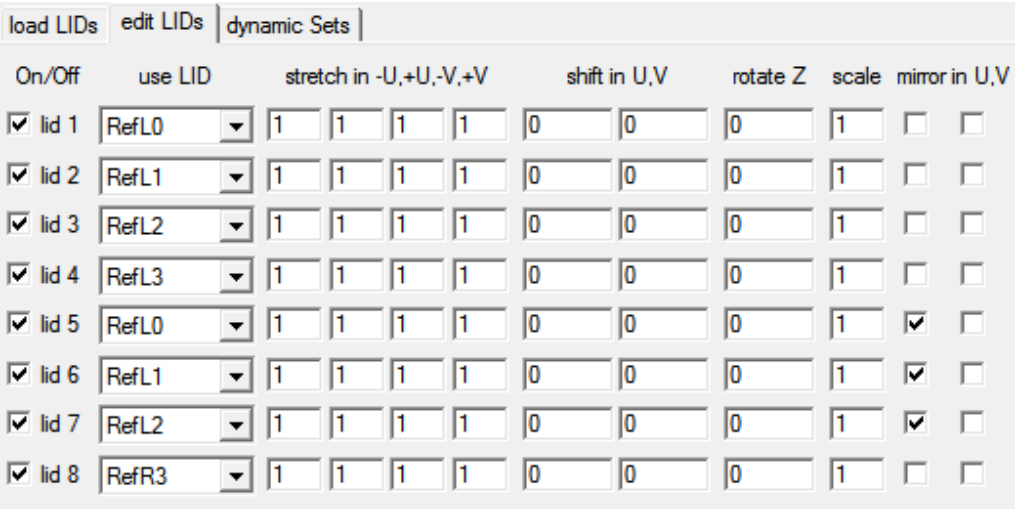


Figure 10.5: The LID Editor: Edit LIDs

flag: on/off	If checked the respective LID is used, i.e. it is taken into account for calculation the resulting LID; if unchecked not.
pull-down menu: use LID	Select here the LIDs to be used in the calculation (the name of the abbreviation is specified in the load tab of this dialog). One can use the same LID several times in the calculation. In the default setting, there are only 5 different LID files, 3 of them are going to be mirrored in the dialog box, which in total leads to 8 different LID files.
stretch in -u, +u, -v, +v	Stretches/shrinks the used LID in -u, +u, -v or +v direction independently from each other. This stretch is expressed as a factor, which does not affect the original LID, it only stretches the LID for the calculation of the resulting LID. The stretch/shrink is performed relative to the respective axis. E.g. a LID going down to v=-4 and a factor of 0.5 leads to an extent down to v=-2. A LID not having light in the +u section cannot be stretched here of course.
shift in u,v	Shifts the LID in u- and/or v-direction for the calculation of the resulting LID, the units are taken in degree.
rotate z	Rotates the used LID about the z-axis, the units are taken in degree.
scale	Scales the intensity value of the used LID.
mirror in u,v	If checked, the used LID will be mirrored for the calculation, this can be done in u- and/or v-direction independently.

The following two figures show the five single LIDs loaded (upper one), they are combined to achieve the resulting LID (lower one). The lower picture shows both the usual view (upper part) and the bird's eye view (lower part). The coordinate systems of all five LIDs are set to identical values (u_{min}, u_{max}, v_{min}, v_{max}) to visualize the position and orientation of each one.

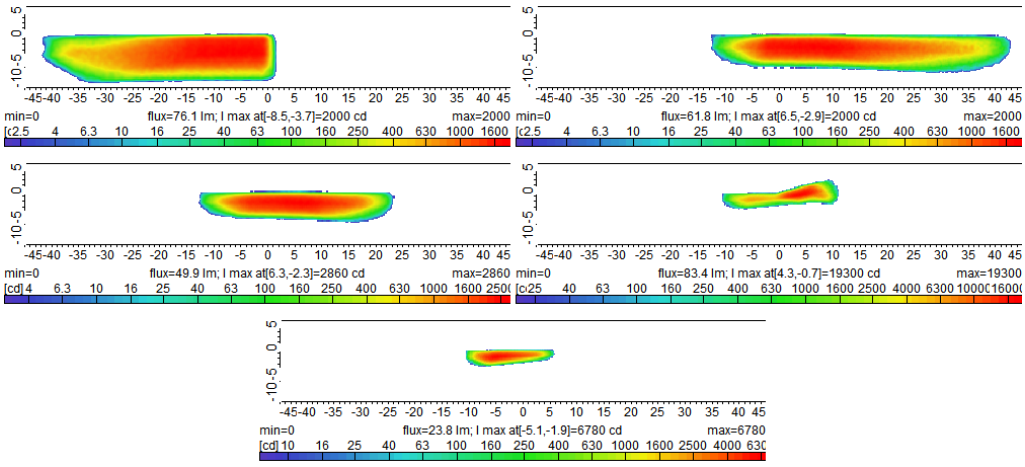

flux=76.1 lm; I max at[-8.5,-3.7]=2000 cd
flux=61.8 lm; I max at[6.5,-2.9]=2000 cd
flux=49.9 lm; I max at[6.3,-2.3]=2860 cd
flux=83.4 lm; I max at[4.3,-0.7]=19300 cd
flux=23.8 lm; I max at[-5.1,-1.9]=6780 cd

Figure 10.6: The source LID files

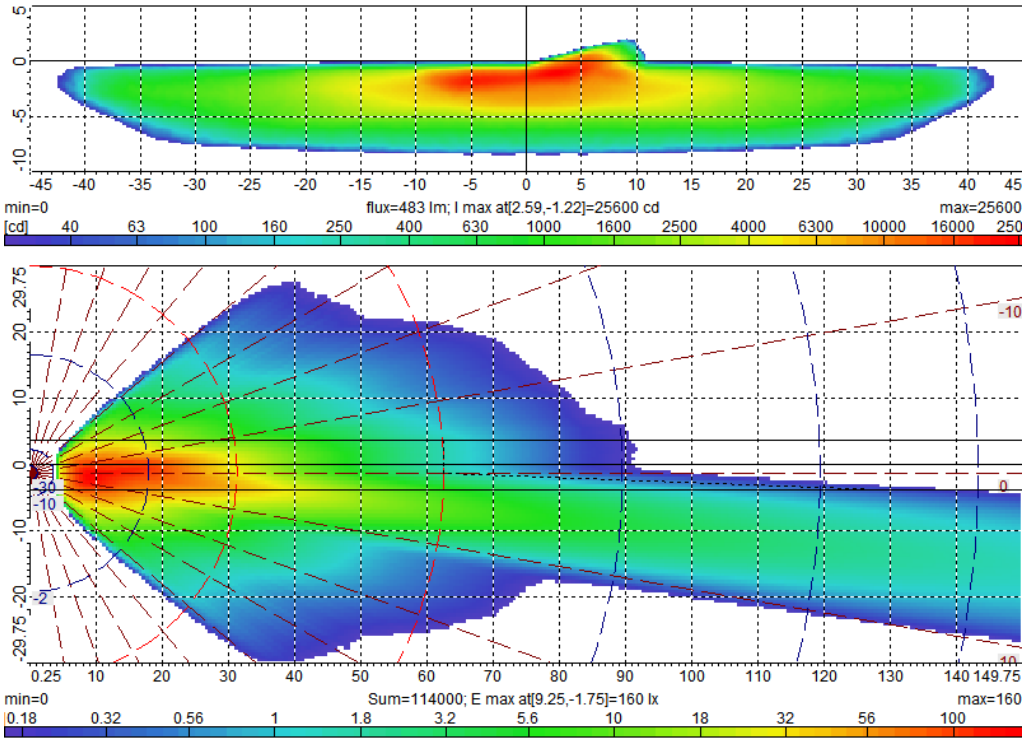


Figure 10.7: The resulting LID

10.4. Dynamic LID Editor

10.4.1. Introduction and Principle

The LucidShape LID Editor also can be used for dynamic LID files. While the dimensions one and two define the intensity over the angular range (u+v), dimension three can be any physical parameter. An additional dimension is introduced when the number of sets is larger than 0.

LID editing of dynamic adaptive systems is one possible application of LucidShape's LID editor. The LID editor supports the creation and interpolation of 3D LID files. For example, one can morph different LIDs into each other. Possible applications for the LID editor of dynamic adaptive systems are AFS bending light or the design of different kinds of wider or narrower high beam pattern.

Light data and/or any other arbitrary distributions are normally displayed in LucidShape over a 2D UV angular parameter range. 3D data in turn holds a third parameter, which can be any arbitrary geometrical or physical parameter, like road curvature. We call this third parameter w. In the UVData views one can navigate through this uvw-data by pressing the right mouse button. While LucidShape supports the handling of 4D data, the LID editor is designed to work with 3D data only.

Individual 3D data consists of sets. Each set describes a sub-function of motion, like the movement of a hotspot or the stretching or compression of a LID. Thus, each set describes a certain dynamic along the parameter w. The data in each set can also be interpolated, either by parameter, i.e. w, or pixel-wise.

10.4.2. Example

After the application is set, press 'apply' and arrange the different LIDs by using the "Window/Tile horizontal" menu item. The picture just below shows the three source LIDs, the resulting LID (lower left part) and the dialog (right part). Again the coordinate system is identical in all LIDs to visualize the position of them.

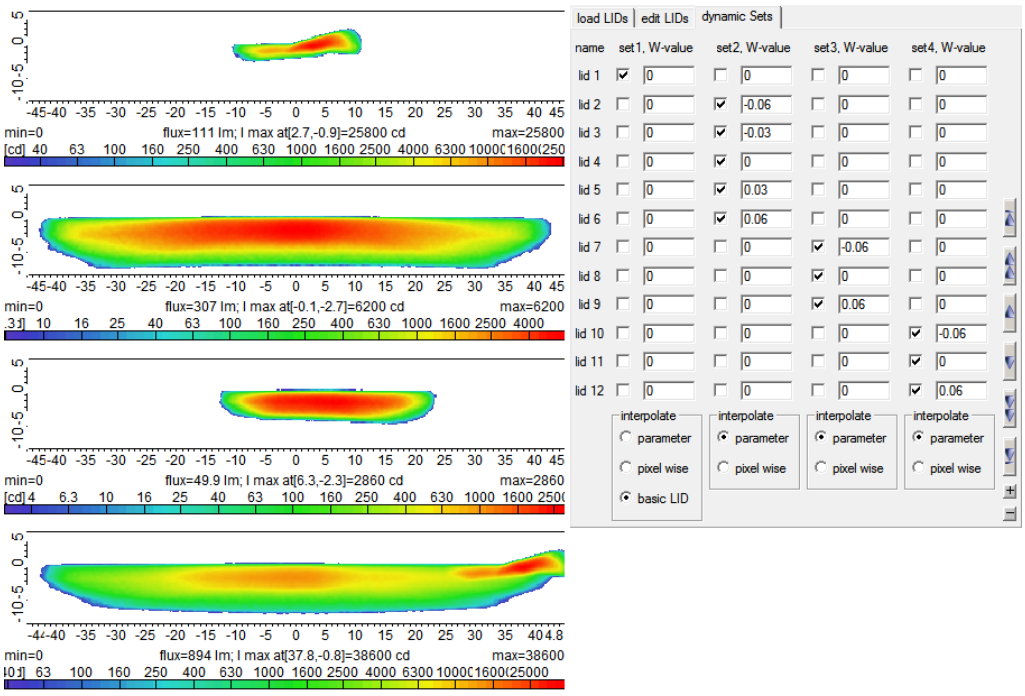


Figure 10.8: The LID Editor and the source LIDs, the sum is the undermost picture

Now, we are going to investigate the difference of 2D and 3D data. Move the mouse into one of the source LIDs, press the right mouse button and open the option "navigate in 3D/4D data set". Because this is a 2D LID file, the arrow buttons are disabled, i.e. there is no third dimension in this LID. Now move the mouse into the resulting LID and open this option again. Now, the arrow buttons can be used to scroll through the data.

Press the play-button and follow the dynamic beam pattern. The w-value corresponds to road curvature, and different w-values show different beam pattern.

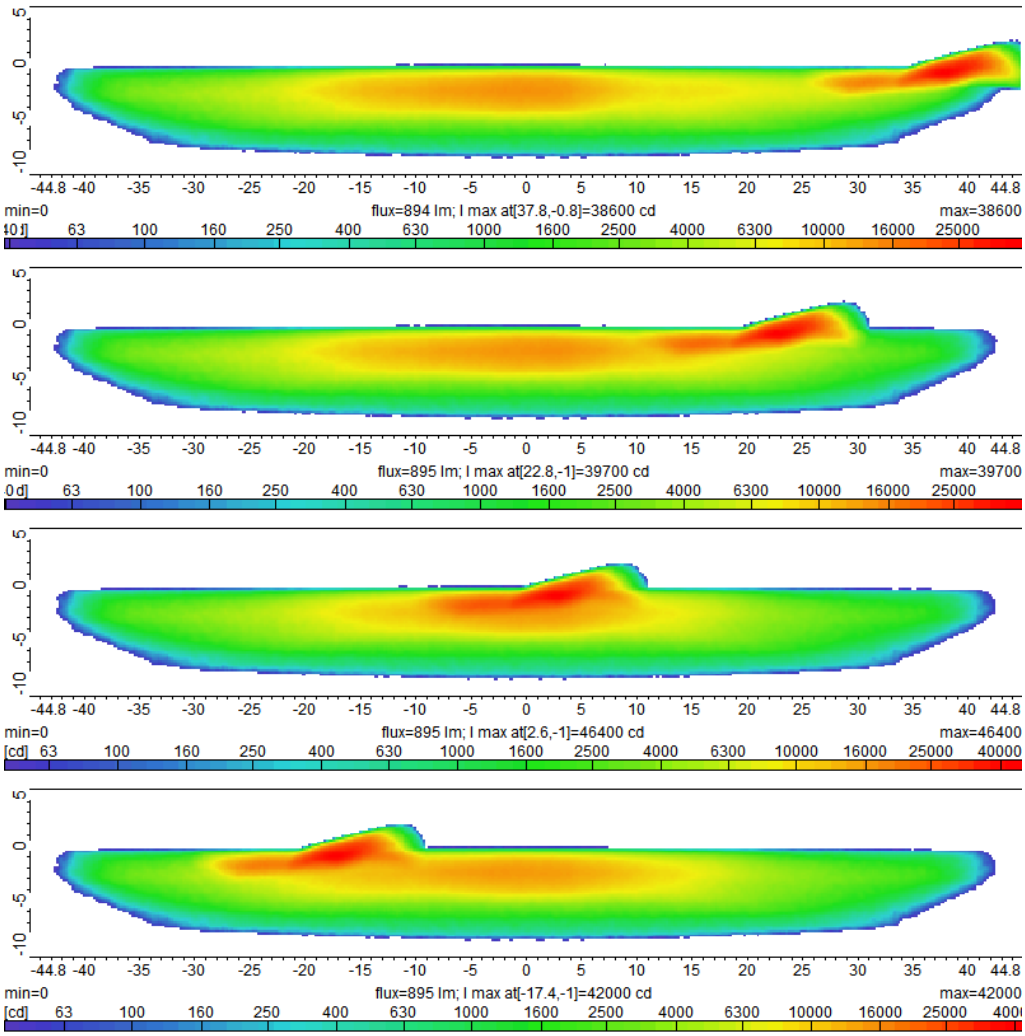
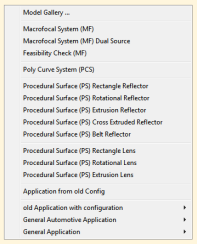


Figure 10.9: Beam Pattern, from the top to the bottom: w=-0.06, w=-0.03, w=0, w=+0.03

Now, in this concrete example, the third parameter w becomes the quantity 'road curvature'. The curvature is the curvature of a road, defined as one divided by the curve radius.

Chapter 11. Applications

The LucidStudio Applications menu offers a collection of very specific tasks and setups, merely for automotive lighting purposes. Please refer to the chapters *Geometry* and *FreeForm Geometries* which describes this collection in detail.


Model Gallery ...
Macrofocal System (MF)
Macrofocal System (MF) Dual Source
Feasibility Check (MF)
Poly Curve System (PCS)
Procedural Surface (PS) Rectangle Reflector
Procedural Surface (PS) Rotational Reflector
Procedural Surface (PS) Extrusion Reflector
Procedural Surface (PS) Cross Extruded Reflector
Procedural Surface (PS) Belt Reflector
Procedural Surface (PS) Rectangle Lens
Procedural Surface (PS) Rotational Lens
Procedural Surface (PS) Extrusion Lens
Application from old Config
old Application with configuration
General Automotive Application
General Application

11.1. General Automotive Application

11.1.1. Static Road Simulation

This application simulates a static road driving situation. It sets the lamp and sensor properties for a static road simulation and checks the light distribution of automobile headlamps on the road. (The sample file can be found within the installation at ..\samples\automotiveLighting\roadSimulation.do.)

In contrast to the rest of LucidShape, the coordinate system used here is almost identical to the one in LucidDrive. The origin (0,0,0) of the coordinate system is located at front of the car on the centre line on the road. The x-axis points straight ahead towards the measuring wall and is the main light propagation direction. The y-axis points to the left and the z-axis upwards. So the default mounting positions of the two headlamps with (0,-1.2,0.625) and (0,-2.4,0.625) mean that both lamps are mounted directly in front of the car (at x=0), the left beam at y=-1.2m, the right beam at y=-2.4m (i.e. the car is located on the driving lane in a right-hand traffic) and 0.625m above the road.

At the very bottom of the main dialog one is able to change the number of aimings, lamps and sets respectively. To apply the change of these parameters, press *Update*.

In the first part of the dialog the aiming or mounting position of a headlamp can be specified, this specifies the position on the car/motorcycle where are lamps are attached. As default there are two entries, one for the left beam and one for the right beam. E.g. to enable the above mentioned combined LID file, increase the number of aimings to three and press *Update*. Next name this new aiming and specify the mounting positions of it (that would be (x=0,y=-1.8,z), with z depending on the vehicle). Now it can be chosen among the other aimings in the dialog part *headlamp beam pattern* below.

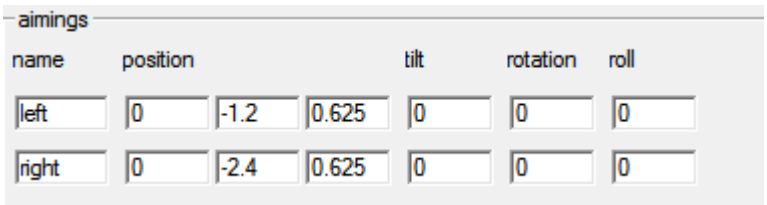


Figure 11.1: The Aimings Dialog Part

name	The name of the "aim selection" as shown in the lamps dialog box below. The default names are left, right and center.
position	The x-,y-,z-coordinates of the lamps.
tilt	The tilt angle (in degree) of the lamps in vertical direction, a positive value points upwards.
rotation	The rotation angle (in degree) of the lamps in horizontal direction, a positive value points to the left.
roll	The roll angle (in degree), e.g. the inclination of the lamps due to the inclination of a motor cycle in curves.

In the next part of the dialog the single lamps with their beam pattern are listed. One can add, remove and modify these pattern (the number of entries is controlled via the value *lamps* at the very bottom of the complete dialog). Each pattern gets a name and has a LID file attached. The aiming of this pattern can be selected among the above specified mounting positions. Additional the color of each pattern can be modified if desired.

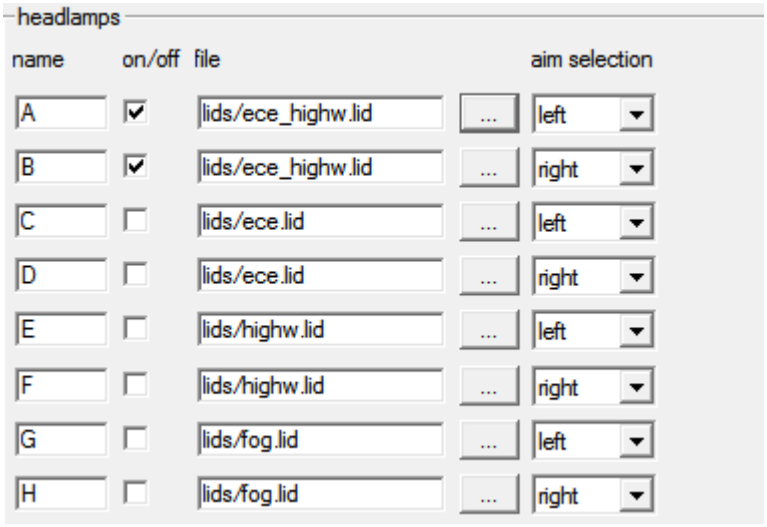


Figure 11.2: The Headlamps and Miscellaneous Dialog Parts

name	Sets the name of the lamps.
file	Via the button '...' the path and the name of a LID file (light intensity distribution) is selectable.
aim selection	Specifies the direction the lamp aims to (as declared just above).

In the part *lamp sets* two or more lamps can be combined to a set (to have a direct access to them, e.g. in a running simulation). Again the number of entries can be altered at the very bottom of the dialog ('sets'). In the default setting there are four sets, low+high, low, high and fog. It is able to mount several lamps or sets onto one car and then turn the respective light on/off via the dialog *LampSwitch* (described below).

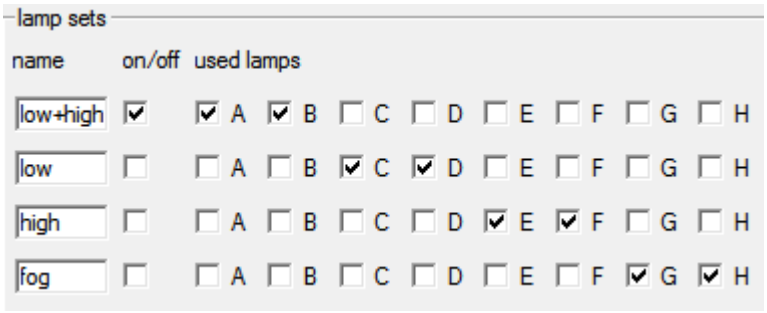


Figure 11.3: The Lamp Sets Settings

The next tab allows to modify the sensor settings.

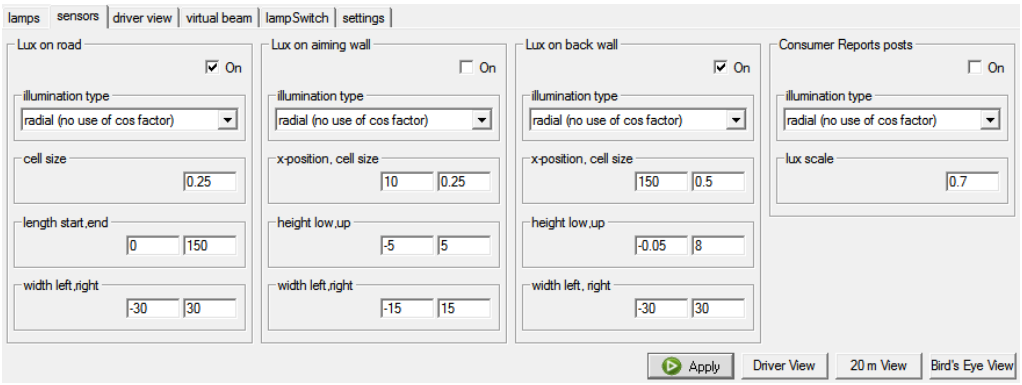


Figure 11.4: The Sensor Settings

flag: on	En- or disables the respective sensor.
flag: Consumer Report posts	En- or disables the appearance of the CR posts on the road.
illumination type	Changes the method the sensor detects the light. For a detailed explanation of the single types and in which cases the use of the cosine correction is recommended, please refer to *LucidTechnical.pdf* within the LucidShape installation.
cell size	The sensor's cell size in [m]. A smaller value increases the resolution and decreases the blocking.
length/width	The length respectively the width of the sensor, both in [m].
lux scale	This factor allows to take a global loss in account (like e.g. absorption etc.).
button: Apply	If pressed, a simulation with the current settings is performed.

The tab 'driver view' allows to set the driver's position and create a UVData diagram with a luminance image [cd/m^2] which is what the driver would see on the road. They also set the field of view.

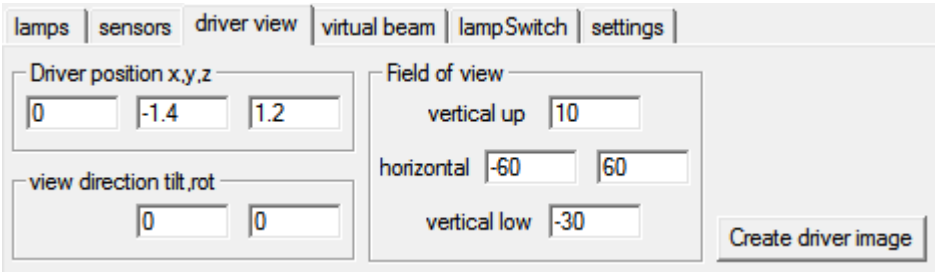


Figure 11.5: The Driver View Settings

driver position x,y,z	Sets the position of the driver.
view direction tilt, rot	Sets the driver's view direction (tilt and rotation) in degrees.
field of view	Specifies the field of view to be shown in degrees (horizontal and vertical).
button: create driver image	If pressed, a UVData diagram for each beam type is created and updated.

The tab 'virtual beam' allows to modify the virtual light source position and LID properties for the reverse sensor light calculation. It creates a "virtual" beam pattern which could have been used to create the illumination in all sensors.

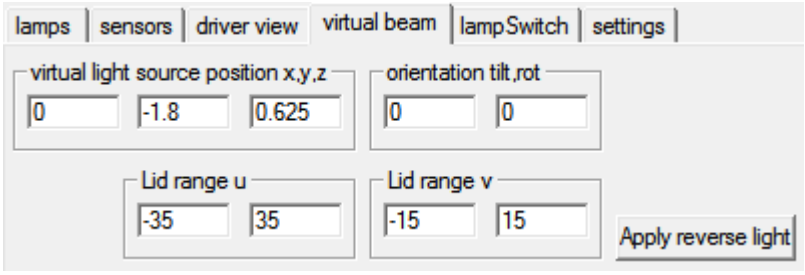


Figure 11.6: The Virtual Beam Settings

virtual light source position x,y,z	The position of the virtual light source in [m].
orientation tilt, rot	The orientation (tilt and rotation) of the beam of the virtual light source in degrees.
LID range u,v	The range of the LID in the candela sensor in u and v.
apply reverse light	Applies the reverse sensor light method. It creates a Candela sensor and updates the LIDData.

The tab 'lamp switch' allows to switch or toggle between the formerly created lamp sets. After one has switched on or off a lamp set, the shown luminance automatically is scaled to distribute the new total light flux onto the complete monitor range (from black to white). Thus different sets with different light fluxes may be shown with the same luminance on the monitor, but with different min/max values for this range. To avoid this automatic scaling, open the sub-dialog *Road Properties* in the tab *settings* and check the flag *fix range*.

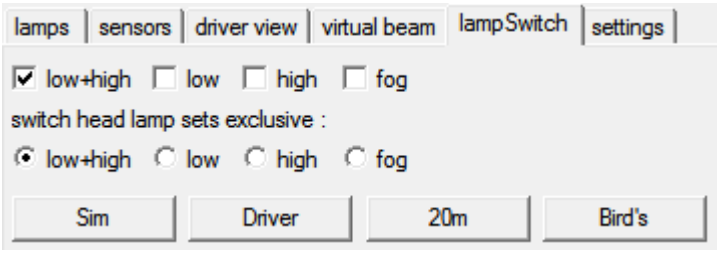


Figure 11.7: The Lamp Switch Settings

In the last tab some general settings may be done. The upper part let you define different positions of the driver, the 20m view and the pedestrian view. The number triple *background color* defines the 'color' and the brightness of the sky and thus the ambient lighting. In case all three values are equal, with increasing values the color goes from black over grey to white. Emphasising a single value will produce a slightly red or blue sky, depending on the user's demands. The lane width allows to alter the width of the driving lane, applied when pressing 'set'.

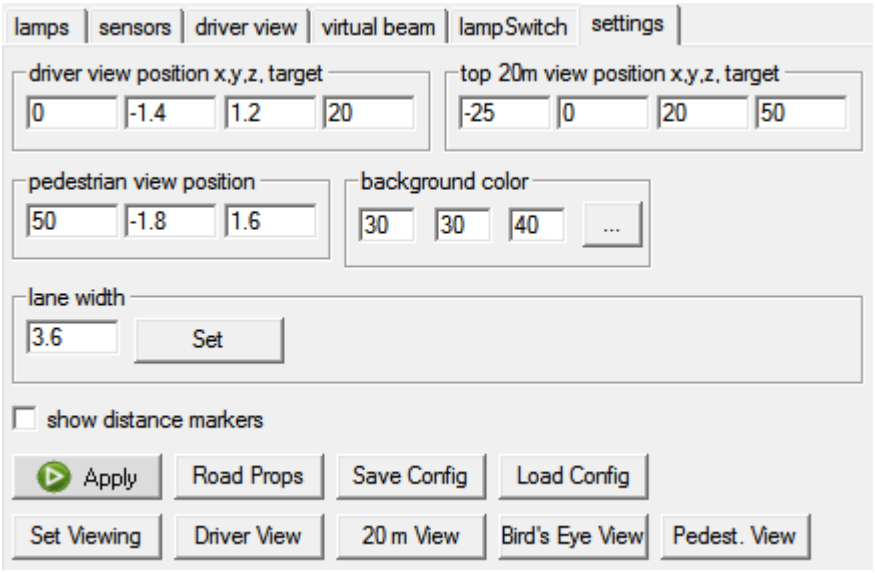


Figure 11.8: The Settings Tab

A 'short' distance to the target will point the aim of the driver's view downwards, this will show more foreground light and raises the horizon towards the top of the window. A value of 1000m or more will place the horizon vertically in the centre of the driver's view.

Some explanatory notes:

- In order to be able to reproduce a measurement with the same settings, it might be a good idea to either save the configuration or record all changed values.
- It is possible to add/import further objects to the scene. Please note that the unit factor in this application is set to [m], so LucidShape objects may need to be scaled to fit into the geometry. After importing an object, mark the GeoView, right-click into it and open the *Surface Display Properties* dialog. Within switch to the tab *lux*and check *TextureLight*, This option will show both the texture and the light on it.
- This application merely projects beams onto the road, it does not perform a ray-trace, so no object will generate a shadow or block the light propagation.
- The stock scene can be shown/hidden via *Menu* → *GeoView* → *Stock Scene*.

11.1.2. Virtual Beam Pattern

This application makes use of the method "Reverse Sensor Light" explained in Section 8.6.2, “Reverse Sensor Light”. The light in the sensor is projected back into a virtual light, so the reverse sensor light is a LID which would be needed to illuminate a scene resulting in identical light distributions in all sensors. One can switch between curve light, fog and low beam light. (The sample file can be found within the installation at ..\samples\automotiveLighting\reverseLight.lus.)

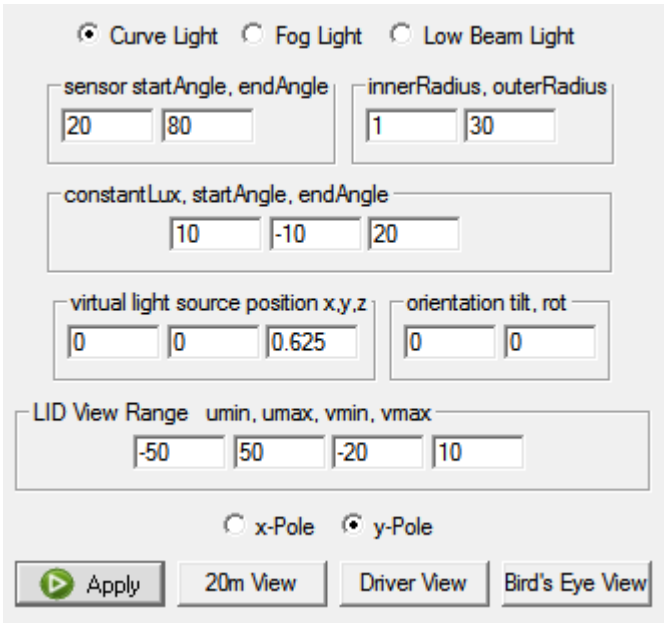


Figure 11.9: The Virtual Beam Pattern Dialog

Please note: Not all of the below described parameters are available for all three beam types.

sensor start/end angle	Defines the size of the sensor via the start and end angle.
inner/outer radius	Defines both the inner and outer radius of the sensor.
alphaF	Sets the width of the low beam sensor for the asymmetric finger.
constantLux, start/end angle	Presets the lux sensor with a constant value. Start and end angle determine the angle range of this offset.

virtual light source position x,y,z	The position of the virtual light source, i.e. the mounting position of the real emitter later.
orientation tilt, rot	The tilt and the rotation angles, i.e. the horizontal and the vertical turn.
LID View Range	Defines the UV-range of the LID via the four parameters u_{min}, u_{max}, v_{min}, v_{max}.
radio button: Pole	Sets the angular type of beam pattern (x-pole = Type B / y-pole = Type A). Try both types and see the different shapes of the beam pattern, even if the LID is the same.
button: apply	Applies the changes into the model.

The last three buttons '20m View', Driver's View' and 'Bird's Eye View' toggle between the respective view position and direction.

11.1.3. License Plate

This is an application for the license plate sensor, containing the sensor with the test points itself (as accessible via Menu → Sensors → License Plate Sensor) and one or more light-sources. The only new parameters in this application dialog below are the options and parameters for the light-sources, that is the number and the exact positioning of them.

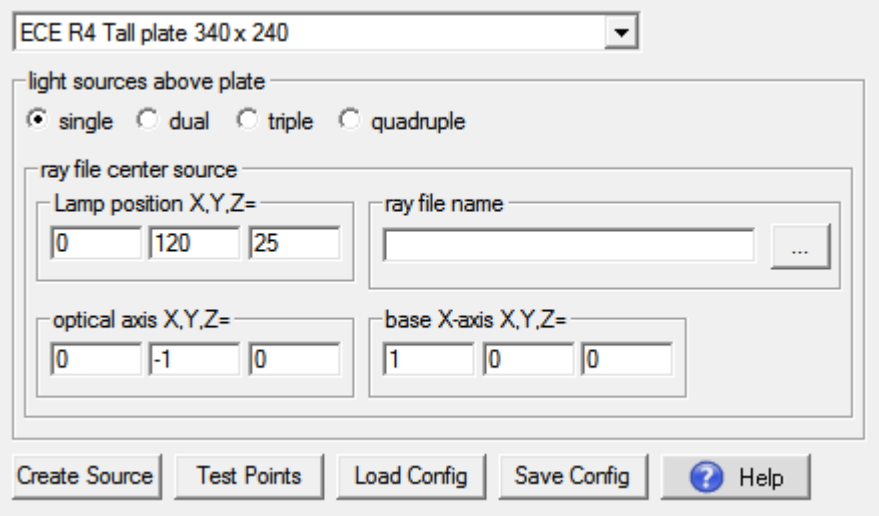


Figure 11.10: The License Plate Application Dialog

radio button: light sources above plate	Determines the number of the light-sources to be created. For each one there will be a full set of parameters to control them independently.
ray file center source	By default the light-sources are evenly distributed; of course the position, the orientation (both x- and z-axis) and the ray file to be used can be altered.
button: create source	After having changed the number of sources or their parameters, press 'create source' to apply the changes onto the model.

button: test points	If pressed, the currently selected test table is applied onto the test point sensor, the result is shown in the MessageView.
button: load/save config	If pressed, the current configuration can be loaded (saved) from (to) HDD.

11.1.4. Retro Reflector

This application is an example for the retro reflection in combination with test tables. It creates a setup with a retro reflector optic, a light source and sensors. The retro reflectance can be checked for different inclination angles according to ECE and SAE regulations. Light from the disk emitter will be reflected from the sample and measured with a sensor. The sample material can be inclined in horizontal and vertical direction. (The sample file can be found within the installation at ../samples/RetroReflex/retroReflector.lus.)

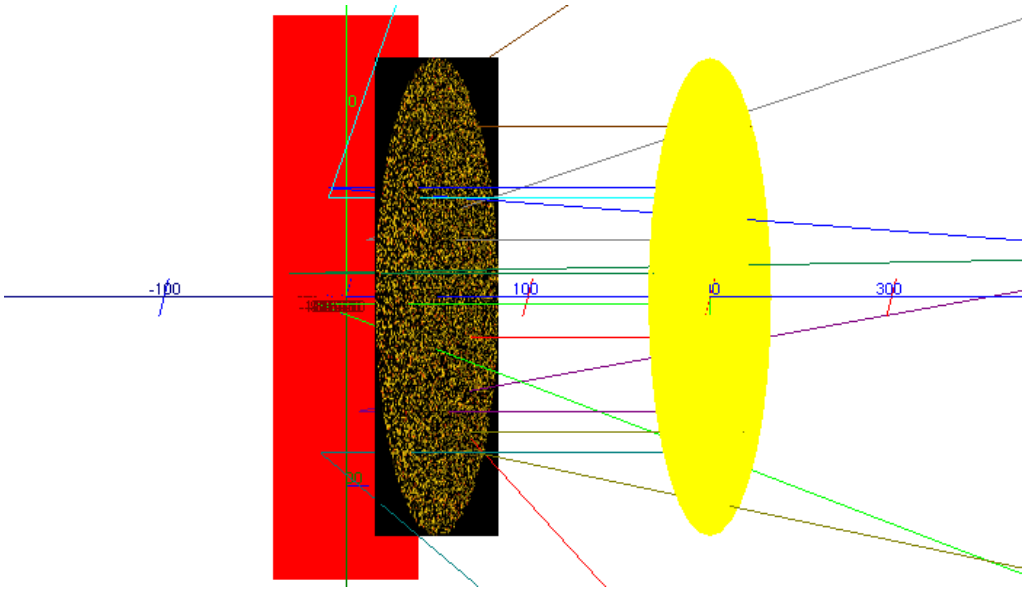


Figure 11.11: The Retro Reflector Sample Model

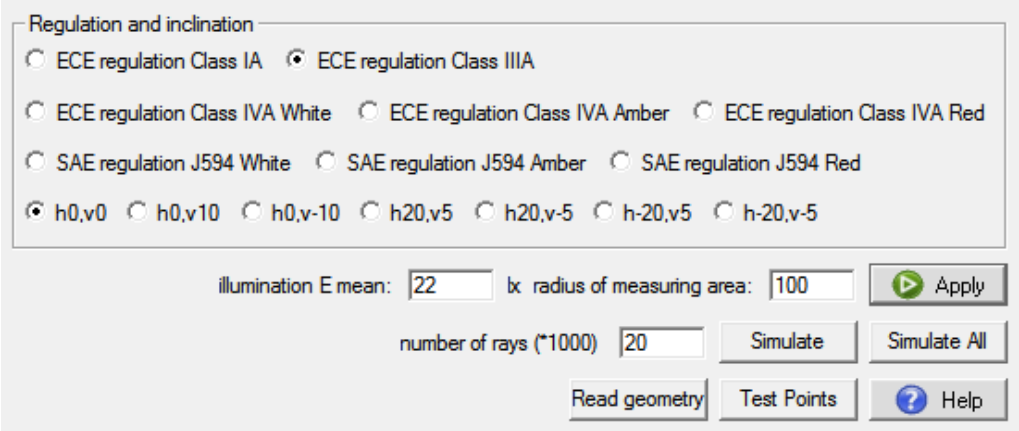


Figure 11.12: The Retro Reflector Dialog

regulation and inclination	In the first three rows, the user can select among different ECE and SAE regulations. The fourth row determines the inclination angle of the sample, h is the horizontal and v the vertical angel, respectively; the units are in degree.
illumination E_{mean}	Determines the mean illumination, given in [lx]. This may be any value, but 22 lx is a common value in practice. The result in the Candela sensor is no longer real candela, but ratios of mil-liCd/E_{mean}, thus no scale is needed in the test table.
radius of measuring area	Determines the radius of the measuring area in [mm].
*number of rays (*1000)*	The number of rays to be simulated in the ray trace, internally multiplied by 1000.
button: apply	If pressed, the changes are applied onto the model.
button: simulate all	If pressed, automatically all available inclinations are set and simulated, each result is saved in an own sensor.
button: read geometry	If pressed, a file selection window opens. Switch to the folder of the sample file and select either the reflective or refractive sample shape.
button: test points	If pressed, the retro reflectance is checked accordingly to ECE and SAE regulations. For two different viewing angles, the measurements are compared with the regulations in the range -20,20 degrees in horizontal direction, and -10,10 degrees in vertical direction.

11.2. General Application

11.2.1. Aircraft Lighting

This application allows to estimate the required beam pattern of aircraft lights. (The sample file is located at ..\samples\aircraftLights\setup.lus.)

This application makes use of the method "Reverse Sensor Light" explained in Section 8.6.2, “Reverse Sensor Light”. In our scenario the aircraft wing, the engine and the logo on the fin are required to be illuminated with a constant value of 10 lx. So the task now is to estimate the required beam pattern of two headlamps which illuminate these aircraft areas exactly with the wanted amount of illumination.

The file setup.lus creates a scene consisting of a wing with two engines, an aircraft hull and a rear fin. The wing, the engines and the fin are covered by a Lux-Sensor with LIDData which is filled with constant 10 lx in each cell. We then calculate the required light distribution for two light positions. One is placed on the hull near the wing, the other is placed near the rear fin. The LIDs can be examined in the respective UVView and are updated automatically as soon as you change the positions and press the "Apply" button.

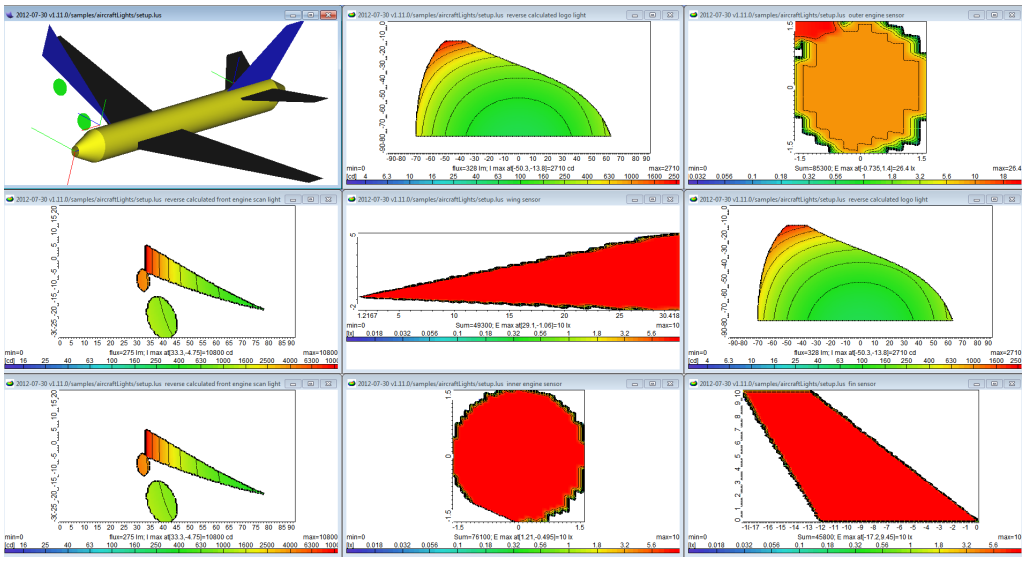


Figure 11.13: GeoView (upper left) and eight UVViews of this application

11.2.2. Test Room for Indoor Lighting

This application simulates the illumination in a test room. (The sample file is located at ..\samples\testRoom\testRoom.do.)

This sample demonstrates the use of the "Gather Sensor Light" method. For a scene where all sensor objects have a far distance to the light sources with respect to the light source size, we may apply the "gather sensor light" method. In this method the light in each sensor is directly gathered from each light source. Here no effects like reflection over other objects are taken into account, only shielding among each other is considered.

In our scenario a test room with 3 walls and a floor is illuminated by a point light source with a LIDEmitter. The number of point light sources in u and v, their distances and the start edge point may be entered. The LIDEmitter uses a pre-calculated or a measured luminous intensity distribution stored in a LID file. The point light sources emit the light according to this LID.

After loading the file you should first switch the geometry view to "Display Light" mode in order to see the illuminance on the walls in 3D! You find the switch in the context menu of the geometry view. Besides the 3D view you may check out the illuminance in the two UVData views, one for the floor and the other for the three walls. Two more candela sensors, one in type C and the other in type A, collect the candela distribution of the source. The results should look the same as your input LID.

You may read in a new lid file via the file selection control, and as soon as you choose a new file the light gathering is started and you see the new illuminance. If you use a blank string for the file name, a LambertEmitter is used instead of a LIDEmitter. You can also alter the light position and the size of the room.

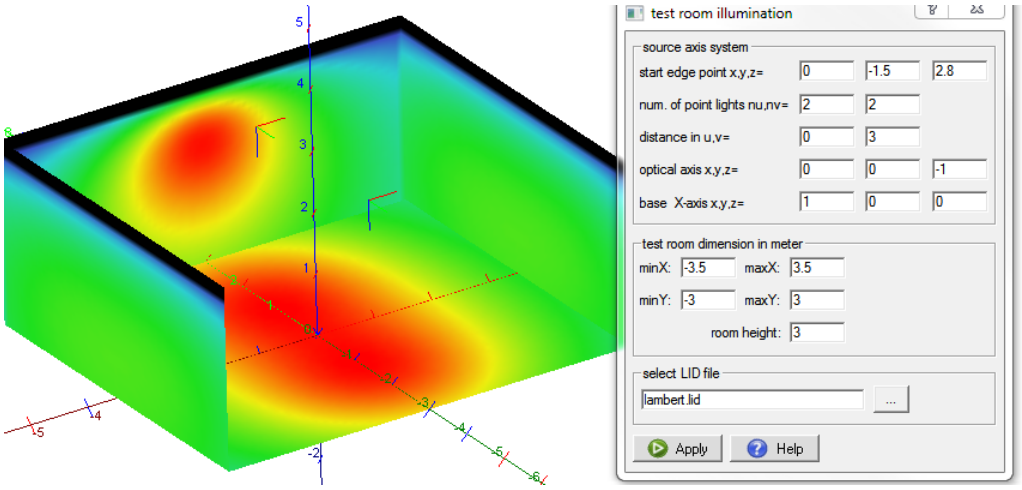


Figure 11.14: The 3D view into the Test Room

11.2.3. Street Illumination

The street illumination application in LucidLite is a versatile tool to display the light distributions from streetlights within a road scene. It consists of two parts, the street simulation and the reverse calculation. The first is the simulation of an imported LID in the defined road scene. The second, however, allows you to reversely calculate from your specified road scene to obtain an ideal LID, which illuminates exactly the defined road surface with a perfectly homogeneous illuminance. From this, you determine the perfect candela distribution for your road scene.

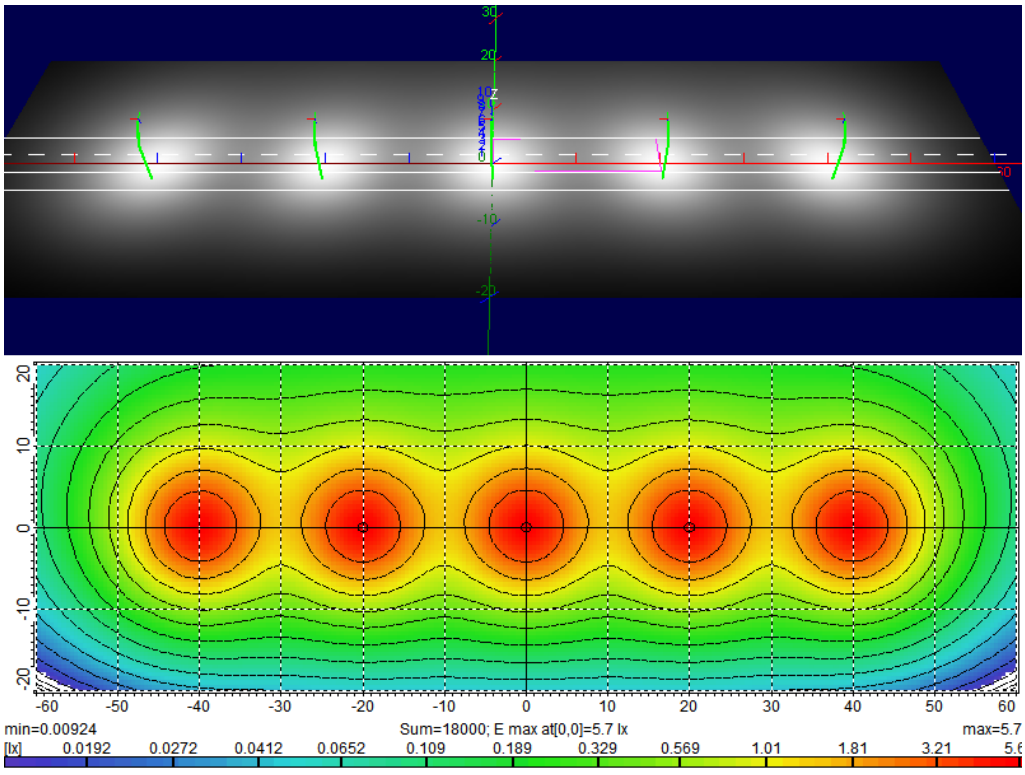


Figure 11.15: Geometry of Street Illumination and LID on the road in [lx]

After the start up of the Street Illumination application, the geometry and UVData windows appear (see figure right above), along with the street illumination dialog window. A sample LID is preloaded to generate the UV data.

For the street illumination application, we speak of a length as the direction or a distance which is oriented along the road, following the driving direction. Consequently, we speak of a width, if a direction or length runs across the road, perpendicular to the driving direction.

Road Types

For street lighting applications, it is in some cases not sufficient to create sufficient illumination, but it is also required to provide a certain amount of luminance from the roadway surface. The results from the according legal requirements. The European regulations of DIN EN 13201 for example, demand minimum values for the average road luminance for the classification of streetlights within the ME/MEW classes (medium to high velocity roads).

To calculate the luminance from the incident illumination, the so-called R-tables are used. These contain the reduced reflectance coefficients of different types of asphalt or concrete that are considered representative for common road surfaces. The exact geometries for the luminance calculation are slightly different between the different regulations, and can be found in textbooks on the fundamentals of general and exterior lighting.

In LucidLite, the need for luminance calculation can be satisfied by the choice of the road sensor type. Usually, the illumination incident on the road (lux) is of primary interest. The luminance from the road, however, can also be calculated.

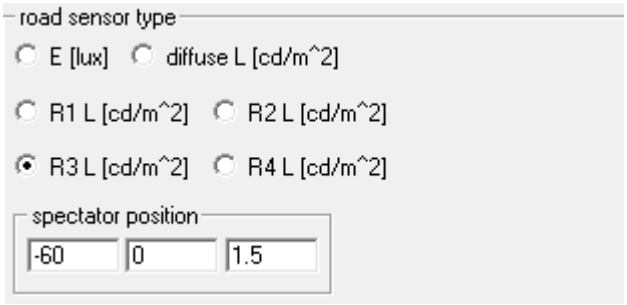


Figure 11.16: Street Illumination Dialog (I)

E [lx]	The standard illuminance sensor.
diffuse [cd/m²]	Perfect lambertian reflectance, diffusely reflecting road luminance.
R1	Mainly diffuse road surface, but of rather high albedo, e.g. concrete
R2	Diffuse/specular mixed darker appearance, e.g. rough new asphalt.
R3	Slightly specular darker asphalt, e.g. old asphalt with worn off roughness.
R4	Rather specular road type of smooth surface, e.g. very fine texture.
spectator position	Determines the coordinates from which the light shall be observed. This has to be set carefully, because the viewing is only valid if looking from the driver view, i.e. on the driving lane.

Upon selection of another road type (R_i), the partial BRDF of this specific asphalt is also shown in a separate uv-data. Remember, these are based on measurements and show the dependence of the polar angle of incidence onto a point on the road and the lateral angle towards the driver position.

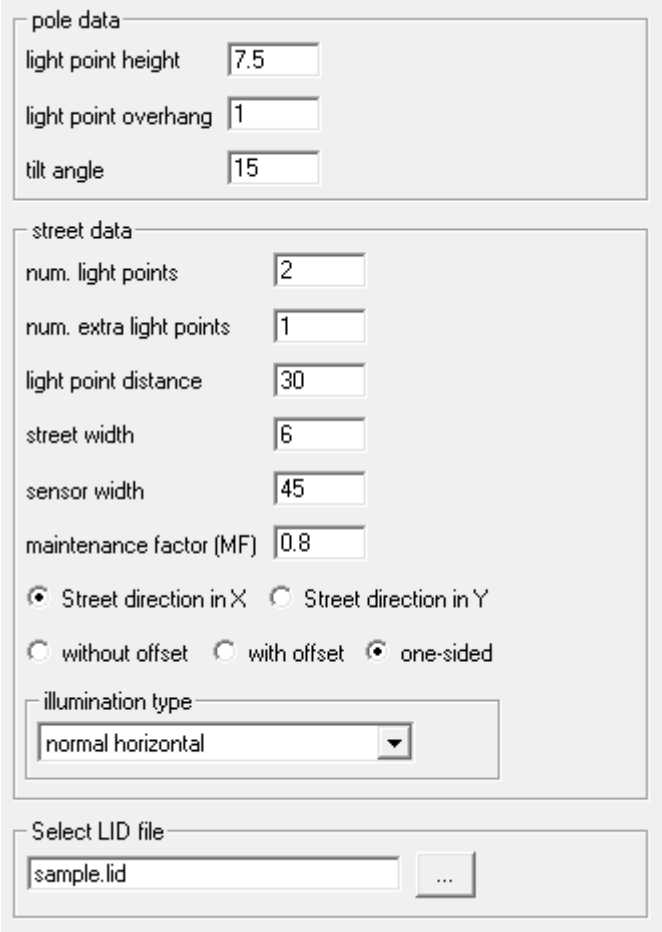


Figure 11.17: Street Illumination Dialog (II)

light point height	The height of the light source (considered as a point) above the road surface, taken in [m].
light point overhang	The horizontal distance of the (point) light source to the sideline of the road and perpendicular to the road length, taken in [m]; a positive value means that the source is above the street, a value equal to zero means that the source is exactly above the side line and a negative value means that the source is places above the sidewalk.
tilt angle	The angle between the vertical z-axis and the light-source's main optical axis, the orientation in which the lamp has been measured vs. the orientation in which the lamp is being installed above the road; dimension of angular degree [°].
# light points	Determines the number of identical lamps along the street.
# extra light points	The extra light points are additional lamps, placed before the centre of the axis system (i.e. in negative x-/y-direction). To have a quite realistic result in the measurement of the complete street,

	there should be at least two extra lamps (depending on their directional characteristic).
light point distance	Determines the distance between two lamps along the chosen axis in [m].
street width	Determines the width of the street from sideline to sideline, taken in [m]. This also influences the position of the light poles via the light point overhang.
sensor width	Determines the width of the sensor perpendicular the street, this is to investigate the light brought to the pavement (and beyond) or to ignore that (i.e. to concentrate on the driving surface itself).
maintenance factor (MF)	The MF is a weighting factor, that represents the reduction of the luminous flux from a light source when it has reached the end of its effective lifetime, i.e. the end of its maintenance cycle. At this point, luminaire replacement and cleaning of the optics is due. The effect of the MF is not shown in the LID, it is only applied when executing the classify function.
radio button: direction	Typically, the street direction ("along the road") is specified as the x-axis of the global coordinate system. The y-axis is therefore across the road. In the U.S.A., however, the y-axis is usually denoted along the road and x-axis across. You can switch this definition for convenience. Please note: If the direction has been changed, the spectator's coordinates have to be adjusted!
radio button: offset	Choose the pole pattern to be two-sided symmetrical (without offset), two-sided alternating (with offset) or one-sided only.
pull-down menu: illumination type	Choose the desired type of illuminance measurement among: Normal horizontal (standard), radial, vertical, half spherical, cylindrical. (To have a detailed description about theses types, please refer to the Technical Reference *LucidTechnical.pdf*.)
select LID file	Choose the file of the streetlight LID to be simulated in the road scene.

Testing the Streetlight

According to the European regulations on street-lighting (DIN EN 13201), there are calculated photometric quantities defined as figures of performance. Please note, that these regulations differ between countries of the European Union, as there is no common international agreement on the selection of illumination classes. For example, the English 13201-2 (E) equals the German 13201-3 (D).

For measuring these quantities on a street, typically a test field between two streetlights is defined. It is filled with a number of test points in rows. Their number is 3 across the road and 10+ along the road. The exact test geometry can be found in DIN EN 13201-2 (E) (7.1 ff). LucidLite applies the very same test point distribution to comply with the regulations and provide simulations as close as possible to measurements in the street.

Figure 11.18: Street Illumination Dialog (III)

apply	Applies the setting to the model, both the street view and the measured LID are automatically redrawn.
scene	Opens a small dialog, here some graphical elements may be shown or blind out independently to each other (the test-field lines, the sidewalk and the test-points). The test parameters can be selected among DIN (Deutsches Institut für Normung e. V.), IES (Illuminating Engineering Society of North America) and full sensor data.
save/load config	Allows to save/load the current settings into a .luc file.
test now!	Applies the tests on the data obtained from the current scene and the selected sensor type on both lanes, the results are shown in the MessageView.
classify (DIN)	Runs the simulation again and calculates test data for illumination and luminance, TI and SR using a DIN accordant scene with some fixed parameters.
compare	If not done until now, classify is executed. After that, this DIN test is compared to the current settings, the results are shown in the MessageView again.
help	Externally opens this chapter of the manual.
driver view	Changes the point of view in the geometry window to the position of a driver on the right lane and at the beginning of the road area.
20m view	Changes the point of view in the geometry window to a position 20m above the road and with the viewing angle slightly tilted down.
pedestrian's view	Changes the point of view in the geometry window to that of a pedestrian at a position above the left side line at 1.5 m in height.
bird's eye view	Changes the point of view in the geometry window to a position far above the road, viewing down.

The street illumination application offers two testing functions: you can initiate them using the buttons "Test Now!" or "Classify". With only one click, the test results are presented in the Message View.

"Test Now": Remember to hit the apply button after changing some parameters, before you use this function.

The "**Test Now**" function calculates test quantities from the last updated simulation results. The obtained values are those of average, min, max, overall uniformity, and longitudinal uniformity. The selected illuminance sensor type is shown in the MessageView directly above the results to remind you of this important parameter. This function can be applied with all sensor-types available in the street illumination application, also the different luminance types. It uses the full road-width of the test field for illumination sensor data, but only a reduced test-field (right lane only) for the luminance values. The SR and TI are not accessible by the test function.

```
Test point grid: DIN
Illumination Type [lx]: normal horizontal
   +---------+---------+---------+---------+---------+
   |   <E>   |  Emin   |  Emax   |  U_E0   |  U_EL   |
   +---------+---------+---------+---------+---------+
new|    5.69 |    4.54 |    7.46 |   0.799 |   0.837 |
   +---------+---------+---------+---------+---------+
mtd|    4.55 |    3.63 |    5.97 |         |         |
   +---------+---------+---------+---------+---------+
```

Figure 11.19: MessageView display after executing the TestNow! function

The "**Classify**" function runs separate simulations without a graphic representation. These make use of the street parameters you have defined but ignore the number of streetlights. Their number is set to six (in case of a one-sided pole pattern): one light in front of the test field, two flanking it, and three behind it. This configuration satisfies or exceeds the incorporation of neighboring lights from -5xMH to +12xMH along the road for most common mounting heights (e.g. for calculation of the TI).

```
Ehs
(5.3594,4.58873,6.43466,0.856203,0.891594)
Ev
(4.70103,3.30499,5.75112,0.703037,0.656759)
Esc
(3.56189,2.06516,6.12671,0.579794,0.898692)
luminance for TI is (right):0.192738 (left):0.192738
--
--
Test results for performance classification
    +-------+-------+-------+-------+-------++-------+-------+-------++-------+-------+
    |  <E>  | E_min | <Ehs> |Esc_min|Ev_min ||  <L>  |  U0   |  UL   ||  TI   |  SR   |
    +-------+-------+-------+-------+-------++-------+-------+-------++-------+-------+
    | [lx]  | [lx]  | [lx]  | [lx]  | [lx]  ||[cd/m²]|       |       ||  [%]  |       |
    | (min) | (min) | (min) | (min) | (min) || (min) | (min) | (min) || (max) | (min) |
    +-------+-------+-------+-------+-------++-------+-------+-------++-------+-------+
new |   5.69|   4.55|   5.36|   2.07|    3.3||  0.193|  0.772|  0.681||   4.75|  0.784|
    +-------+-------+-------+-------+-------++-------+-------+-------++-------+-------+
*MF |   4.56|   3.64|   4.29|   1.65|   2.64||  0.154|       |       ||   -   |       |
    +-------+-------+-------+-------+-------++-------+-------+-------++-------+-------+
```

Figure 11.20: MessageView display after executing the Classify function (the last line shows the MF effect)

The important quantities of roadway lighting are listed briefly in the following table, along with the important details of their numerical implementation.

min, max	Minimum and maximum values of an illuminance or luminance, respectively.
⟨E⟩	Average value of the illuminance E, averaged over all test points within the test field, it is often also denoted as E_o, $\langle E_h \rangle$, or simply with a macron above.

E_v	The vertical illuminance is measured on the road (test) or at a level 1.5m above the road surface (classify). The minimum value of E_v is supposed to reflect for example the recognizability of road signs
E_{hs}	The half-spherical illuminance is measured at road level.
E_{sc}	The semi-cylindrical illuminance is measured at the road surface level (test) or at an elevated level, 1.5 m above the road surface (classify). Measurements of E_{sc} provide a measure of the human ability to recognize faces under the measured streetlights. This is of higher importance in city areas.
U_o	The uniformity of the luminance from the road is calculated via the quotient of average and minimum of each lane separately. The displayed value (classify) is the smallest value found in either lane.
$U_{E,o}$	The overall uniformity of the illuminance is calculated via the quotient of average and minimum of all test points on the road. *These are not required by regulations, but can assist in design and evaluation.*
$U_{E,L}$	The longitudinal uniformity of the illuminance at the center line of the road (along the road) is calculated via the quotient of minimum and maximum of all test points at the center line. *These are not required by regulations, but can assist in design and evaluation.*
U_L	The longitudinal uniformity of the luminance at the center lines (along the road) of the lanes is calculated via the quotient of minimum and maximum within each lane. Shown value is the minimum of either lane (classify).
‹L›	Average values of the luminance L, averaged over all test points within a lane, sometimes also denoted as L_o or simply with a macron above the L. After using the classify function, the displayed value is the minimum from all lanes.
TI	The threshold increment is a measure of the negative effect on human perception caused by the low angle luminance of street lights that are directly above or not close by the observer. Lights along the road are summarized until their effects reaches a saturation of 2% increment or less.
SR	The surrounding ratio is the quotient of the average illuminance on two outer zones of the road and the average illuminance on the adjacent pedestrian ways ((street L+R) / (surrounding L+R)). All four test stripes are of the same width (i.e. the lane-width, but 5m max.). The applied test stripes are again equipped with the test point matrix (3 x 10+), just like the individual lanes and the complete road are being measured.

The displayed photometric values are those of ‹E›, E_{min}, ‹E_{hs}›, $E_{sc,min}$, $E_{v,min}$, ‹L›, U_o, U_L, TI, and SR. Also, the impact of the maintenance factor MF is shown in the second line of displayed data.

11.2.4. Reverse Street Light Pattern Calculation

The reverse calculation method of the street lighting application allows you to determine a perfect lighting solution for the specified road scene. It makes use of the point source assumption, which yields fairly well due to the rather high distance between light-source and road and the comparably small size of the lamp itself. Besides the road parameters, you have to choose a value of the homogeneous illuminance [lx]. From the calculation, you obtain the candela distribution for the chosen homogeneous illuminance on the road surface (side range times street width). Also, the required luminous flux is calculated.

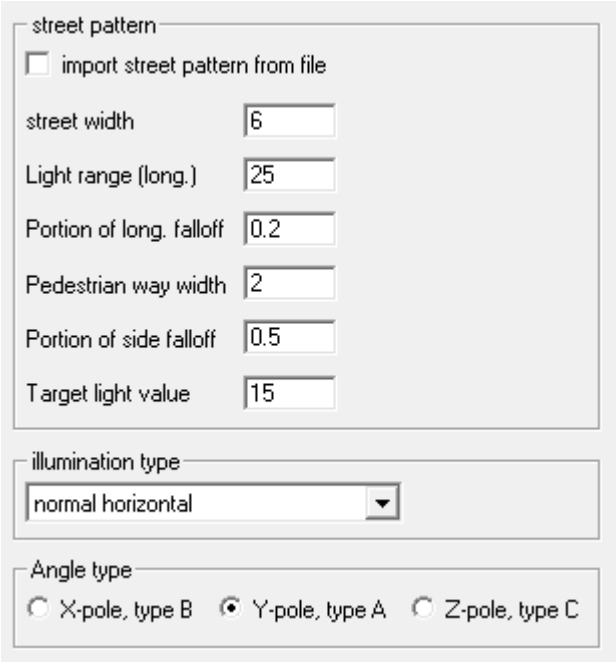


Figure 11.21: Reverse Street Light Pattern Calculation Dialog

Most of the options and parameters are identical to those described in the previous subsection, so only the differences will be explained. The length specifications are either in [m] or normalized to the respective longitudinal/side range.

flag: import street pattern	If checked a self-created .lid file can be imported, this choice replaces two parameters from the dialog.
light range (longitudinal)	Determines the long. extent the light shall illuminate in [m]. (This is quasi the distance between two lamp poles.)
portion of long. falloff	Defines the relative length of a zone of decreasing of illuminance, starting at the border towards the centre of the illuminated roadway (normalized to side range). From the defined position the illuminance will fall down with a constant gradient towards the very end of the side range. (A smaller value leads to a sharper cutoff, but there are two limiting factors: The resolution of the sensor and the assemblability of the lamp).
pedestrian way width	Determines the width of the pedestrian way beside the road surface. This width is added to the street width to calculate the complete extent to be illuminated.

portion of side falloff	Defines the relative length of a zone of decreasing of illuminance, starting at the border towards the centre of the illuminated roadway (normalized to side range). From the defined position the illuminance will fall down with a constant gradient towards the very end of the side range (e.g. 0.5·side-range would be half the side range on either side).
target light value	Determines the wanted amount of light on the middle of the road (outside the falloff, in [lx]).
radio button: angle type	Selects the angle type which shall be used for calculating the LID, i.e. the coordinate system.

The picture just below shows an example, the wanted illuminance distribution (upper part) is given either directly via a .lid file or via the entered parameters; the lower part shows the resulting LID to be 'inserted' in the lamp pole to achieve the specified LID.

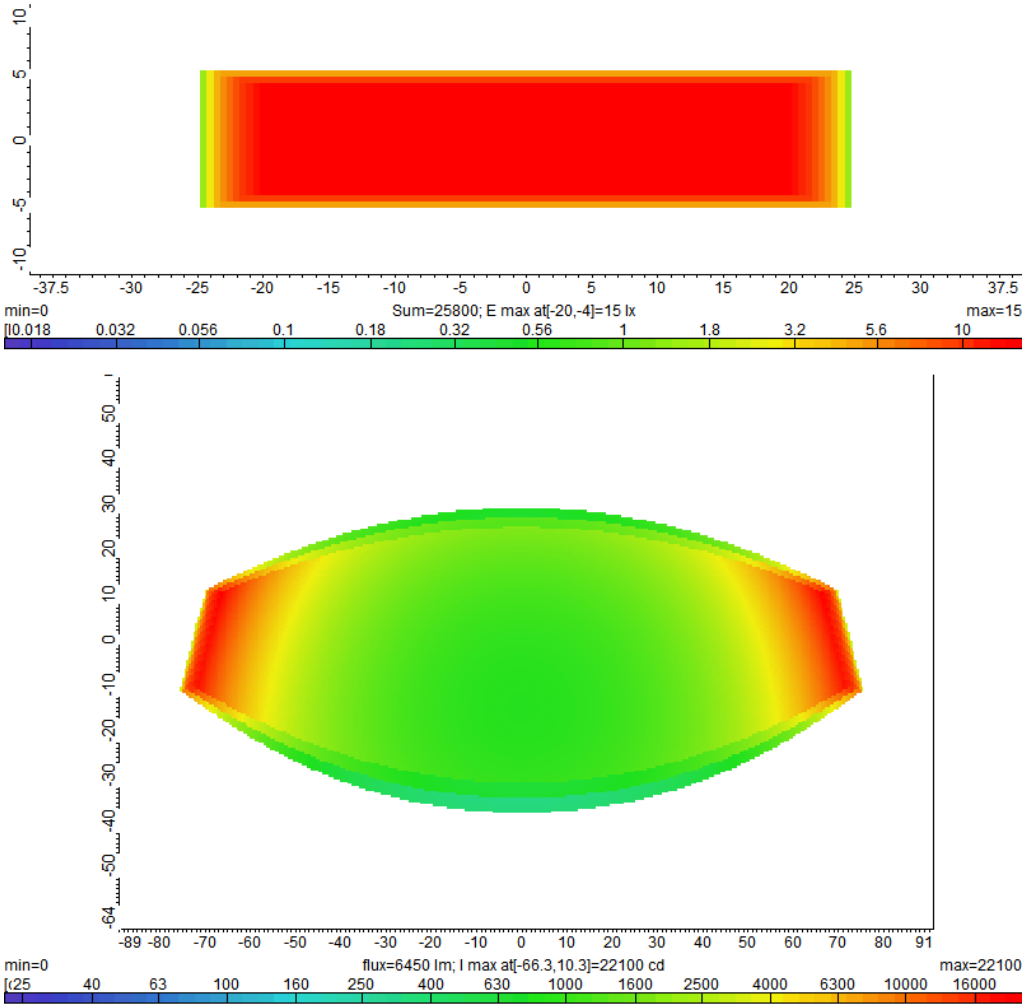


Figure 11.22: Example with the wanted LID on the road (upper part) and calculated LID (lower part)

11.3. Tests

The four tools *Filament Image Viewer*, *Source Position Checker*, *BSDF Maker* and *Sky Light Checker* have fairly similar tasks, furthermore the dialogs do not differ very much. All four tools are used quite the same way:

- At first press "into model", this creates a new experiment, also some specific objects will be created automatically.
- Add some tool-depending objects into the experiment, e.g. in case of the BSDF Maker add a suitable scattering surface or when using the Source Position Checker create a sensor-surface).
- Specify an array of values by determining the respective minimum and maximum values, pressing "run all" will run (i.e. simulate) all possible permutations, automatically altering the geometry to the current set of values.
- The middle part of the dialogue concerning something like "rotation alpha, beta", "rotation theta, phi" or similar are independent to the just mentioned array! This is only for manually testing or adjusting the setup (position and orientation of the light-source) before activating "run all", so these values have no influence on the animation array.
- Do not create an object the dialog will create automatically (e.g. the BSDF Maker creates its own emitter and sensor and will not work properly with self-created ones).

11.3.1. BSDF Maker

This sub-section describes the use of the application *BSDF Maker* which can be found under the menu item 'Test'.

In short: This tool is a rudimentary goniometer-type experiment to create your own BSDF data. You specify a sample material, define scanning ranges and resolutions, start ray-tracing and recording - and the simulated LID is converted into a BSDF dataset. This BSDF can be used as BSDF material for creating a shape with the same scattering behaviour as the original sample.

1. Open the dialog and press 'into model', this creates a new geometry window containing a disk-shaped light-source (directional emitter) and a sensor (type C).
2. Add the sample as an shape into the geometry, it is strongly recommended to have the centre of it at the position (0,0,0) and its normal parallel or anti-parallel to the z-axis. The light automatically is being emitted in either (0,0,1) or (0,0,-1), according to the type of the material (reflective or refractive). Depending on the size of this sample it might be necessary to adjust the size of the emitter. Illuminate the sample with a rather large polar angle, if any light is missing the shape, the emitter's radius has to be reduced.
3. Now the angle range and step-size for both θ and φ have to be specified. Please note that the θ-range has to start at 0°. The ranges determine the array of scanning angles the sample is illuminated from.
4. Press 'reset', the light-source is positioned at the starting position and the given number of rays are simulated. This gives a first impression of the resulting LID.
5. If all parameters have been set, mark the GeoView and press 'run all' to start the recording process. All permutations of the angle pairs are simulated, all these single shots are saved into the three-dimensional sensor.
6. Mark the UVView and press 'make BSDF', this converts the LID data into BSDF data. It appears as a second UVView but with a different name: 'BRDF from Maker'. Mark this new data and save it onto your HDD. (In case the name of the windows does not update automatically, perform a double-click into the GeoView.)

7. The further proceeding now is to create an actor material from this just saved file: Menu → Material → Create Actor Material → 3D BSDF Reflector/Refractor and choose the BSDF file. Create or import a surface, then assign the material via the entry in the TreeView onto this surface, as a result you obtain a BSDF shape.

The number of rays used for each single simulation is quite important, using a too low number results in a high noise level, a too high number noticeable increases the file size of the saved BSDF data. The delta of the angle(s) (thus the angular distance between two shots) should be rather small to increase the resolution of the BSDF data. A recommended delta is about 1° to 3° but not more than 5°.

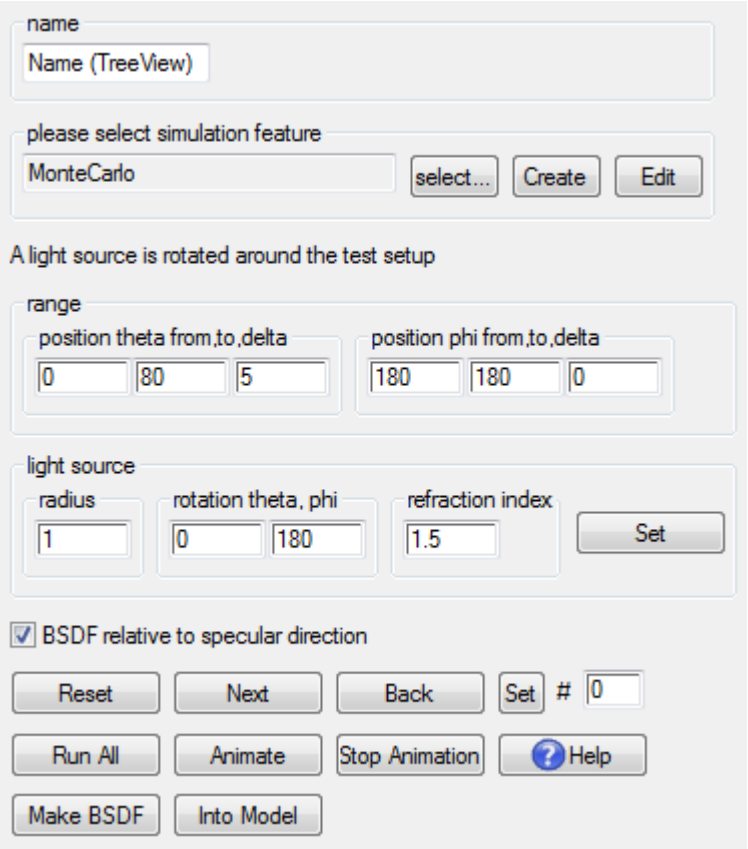


Figure 11.23: The BSDF Maker Dialog

simulation	For the simulation, select a SimulationFeature from the TreeView, create a new one or edit the currently active one.
range	Determines the range to be run through for both the θ (polar angle) and φ (azimuth angle) and the respective delta (step size): $[\theta_{min},\theta_{min}+\Delta_\theta,..,\theta_{max}]$ and analogue for φ.
light source	The radius and the initial position of the emitter, press 'set' to apply changes. This position is independent to the array and can be used to perform a manual simulation at a certain angle (e.g. to check the size of the emitter, see above). The correct refraction index automatically is taken from the model, do not alter it manually.

flag: BSDF relative to specular	This flag has to be checked for reflective material and unchecked for refractive material. In case the BSDF data was created with the wrong setting, you can toggle the data via the context menu and then re-save it (the UVView states *rel to spec* when the data is relative to specular).
button: reset	If pressed, the emitter is moved to the initial position and orientation, additional the single simulation is done.
button: next/back	If pressed, the emitter is set to the next/previous angle pair, additional the single simulation is done.
button: set #	Although the angle pairs form a two-dimensional array, these combinations are treated like a list. If pressed, the emitter is set onto the chosen combination i.e. list entry.
button: run all	If pressed, the complete array of angle pairs is run through, recording the single simulation results in the automatically created sensor.
button: animate	If pressed, an endless loop of simulations is started.
button: stop animation	Pressing this button stops the animation.
button: make BSDF	If pressed, the LID data is converted into BSDF data. Please note: Without this conversion the recorded data cannot be used to create an actor material.
button: into model	If pressed, a rudimentary geometry with a light-source and a sensor is created. (This tool does not work with a self-created sensor.)

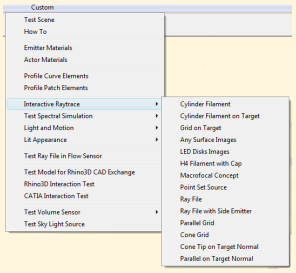
As already mentioned above, do not change any object from the application itself because this will result in data errors!

Chapter 12. Miscellaneous Menus

This chapter describes (mostly in tabular form) miscellaneous menus which do not require a detailed description, or which are described in detail in other LucidStudio documents.

12.1. Custom

The "Custom" menu can be customized to your individual needs. You may extent the menu interface by programming in LucidShell language and setup the interface for your current project. For example, if you work on a project where you have several calculation procedures for each reflector surface, you can make menu entries for each reflector part and enter your reflection angles into your "home made" dialog box.



12.2. Options

12.2.1. Global Settings

This menu entry opens a dialog that consists of seven entries and presents various program settings like paths, CAD data exchange modes etc. The installation process automatically adjusts the respective entries consistently to the structure created on your HDD, so this usually only has to be modified if there were made some changes.

Global Settings

tab: Path 1	In this tab four directories can be set, where to find the respective folders or files, namely the scripts, the classes, the lush.exe (this is not the acronym for LucidShape) and the pdf files.
tab: Path 2	In this tab an external text editor can set, also a temp directory, the path to the company logo, a background image file and the mesh2lug converter tool. (This tool usually is installed directly into the chosen LucidShape folder, by setting the path it can be used in different LucidShape installations.)
tab: Libraries	In this tab the paths to the respective objects can be changed, already set are the default settings according to the structure of the installation.

CAD Transfer

pull-down menu	In the pull-down menu all set axis can be seen and selected.
buttons	The buttons can be used to set, to add, to delete or to list the current axis.
radio button: read CAD via	Chooses the CAD program to be used for reading data.
radio button: write CAD via	Chooses the CAD program to be used for writing data.
radio button: version	Allows to select the version to be used.
radio button: material exchange type	Enables or disables the exchange of materials between LucidShape and Catia resp. Rhinoceros (which do not natively support physical materials).
flag: color and textures	If checked the parameters are taken from the Rhinoceros Render Material library.
flag: log import/export	If checked and performing the respective transfer, all occurring error messages will be saved in this file. It will be created in the respective loading/saving folder, named identical to the model file.

Graphics

radio button: NURBS shading	Choose the NURBS shading mode in the GeoView between either via *OpenGL* or via *tessellation* (tessellates the geometry and displays the shades triangles).
radio button: trimmed surface shading	Choose the shading mode of trimmed surfaces in the GeoView between either *via OpenGL* or via *tessellation*.
surface tessellation parameters 1	These three values determine the appearing in the GeoView and the tessellation used in simulations when using the *Triangle* or *Mixed Mode* simulation method. • min. edge length: This is the minimal length of the triangles' side. This lower limit is needed to prevent too small triangles. • max. edge length: Lowering this value will force the tessellation process to create more triangles, the recreated surface becomes 'smoother'. • max. chord height: Discretization parameter for surfaces inside the geometry, smaller values lead to a finer discretization. This is the maximal allowed distance between the original surface and the triangle created here (also see the sketch below this table).
surface tessellation parameters 2	• max. auxiliary nodes per surface: This is the maximal number of triangles or vertices created when performing a tessellation of the model. This number can be limited to reduce the system requirements, but should be set to 0. Zero means that as much triangles or vertices as necessary are created to fulfil the *surface tessellation parameters 1*. There is a protection included to

	prevent system crashes due to an unlimited number of triangles for bad surfaces. • smooth rotational tessellation at poles: When tessellating a surface near a pole, one single point is a vertex for several triangles. If activated, the pole area is tessellated much smoother. • generate tessellation debug files: If activated, a debug file is generated when performing a tessellation. This can be used to analyse problems.
curve tessellation parameters	This is almost the same as described just above but for curves instead of surfaces, therefor the meaning of the parameters are almost identical.
default line width	Determines the standard width of drawn lines in LucidShape (this does not alter the line itself).
flag: display curves with arrow direction	If checked all curves are drawn with an arrow on it, this allows to see the direction of the parametrization (this might be important if curves are used for trimming).
button: redraw graphics	Applies the current setting to the GeoView in focus.

Dialog

The flags "hide views of ...": If checked the respective object dialog will not show up automatically when opening the model.

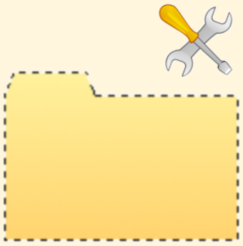
flag: dialog boxes as child views	En-/disables the possibility to move the dialogs outside the LucidShape frame. (For dialogs this option is selectable, views never can be moved outside).
flag: do not remind for save	If unchecked LucidShape will ask for saving all altered models opened.
flag: prompt overwrite existing file warning	If checked, a prompt will pop up every time the 'save' function is used, this is to avoid accidental overwriting of a file.
flag: implicit open all design feature dialogs	If checked all available design feature dialogs in the model are opened when loading it.
flag: implicit execute apply on all design features	If checked, an apply is executed on every design feature after the model has been loaded.

12.2.2. Working Directory for Open/Save

Allows the setting of a default working directory.

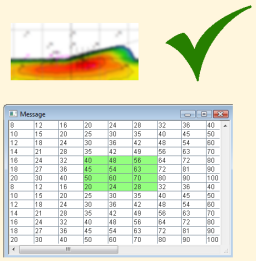
• *Set Working directory for Open/Save* opens a file selection dialog to set the working directory.

• *Use Working Directory* enables or disables a flag that will make all open and save file dialogs start from the working directory.



12.2.3. Editing of LID Test Tables

The edit dialog in this subsection is accessible via "Menu → Options → Edit LID Test Table". It can be used e.g. to alter the provided test tables, to include an own library of regulations or to 'merge' different test tables (e.g. from different countries).

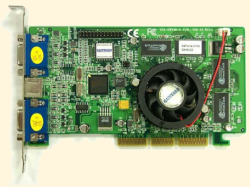


In LucidShape the LID Test Tables are implemented very flexible. Provided tables easily can be altered or deleted and new ones can be created. To ensure this flexibility, all test tables consist of multiple single measurement tasks.

The function of the single measurement tasks, the possible customizations (adding a folder, editing or deleting already existing test tables and creating new ones) are explained in detail in the new separate whitepaper *LIDTestTable.pdf*, located at http://www.brandenburg-gmbh.de/products/lucidshape/downloads/whitepapers/.

12.2.4. Hardware Graphics Accelerator

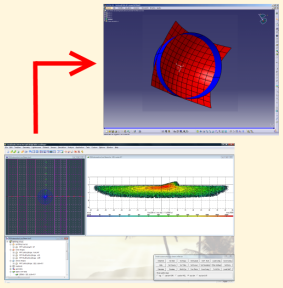
If this flag is enabled, the LucidShape geometry rendering of 3D objects will be supported by the graphics board (if any is installed). Otherwise, 3D objects will be rendered by standard MS Windows routines.



12.2.5. COM Server

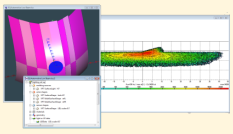
The MS Windows COM technology is used for seamlessly (i.e. without having to read or save data in temporary files manually) exchanging geometry data between LucidShape and external tools like CATIA or Rhinoceros. You must apply *Register this program as COM server* before you can perform any such data exchange, preferably before you start the external tool.

Apply *Unregister this program as COM server* when you are finished with your data exchange (as any COM server needs some system resources).



12.3. Window

This menu presents various common window positioning commands.



Cascade	Cascades all non-iconized view.
Tile Horizontally	Resizes and arranges all non-iconized views horizontally so they fill the LucidShape desktop.
Tile Vertically	Resizes and arranges all non-iconized views vertically so they fill the LucidShape desktop.
Arrange Iconized Views	Arranges all iconized views at the bottom of the LucidShape desktop.
Close All	Closes all open views. In addition, type-specific commands (like "Close All GeoViews") are available for all types of LucidShape views.
Open Message View	Opens the message view, if it is not already open, and positions it at the right bottom corner of the LucidShape desktop.
Open Text Edit View	Opens another text edit view for the current model in focus. Not applicable if there is no model in focus.
Open Tree View	Same like *Open Text Edit View*, but with the TreeView.
Open Geometry View	Same like *Open Text Edit View*, but with the GeoView.
Light Screen View	Same like *Open Text Edit View*, but with the light screen view.
All UV/Light Data Views	Opens all available light data views of the current model in focus.

12.4. Help

Most entries under this menu offer shortcuts to various LucidShape documents. Also, a license check may be performed.

12.4.1. License Check

Lists all licensed modules of the current LucidStudio version in the MessageView (there will be no such output when using the dongle-only version of LucidShape).

12.4.2. Shortcuts to LucidShape Documents

The menu entries in this section offer shortcuts to LucidShape documents. Available documents are

LucidStudio	This document, which is the main document.
Introduction	A short introduction to the main concepts of LucidShape, this is a good choice when starting from scratch.
Tutorial	This entry contains a new series of tutorials. Among others, the topics are the basics of the forward ray trace, materials, actors, sensors, spectral simulation, analysis of the results, mathematical approaches and three main FunGeo concepts (MF, PCS, PS).
Macro Focal Application (MF)	A short additional handbook which describes the concept and use of MF in detail.
Procedural Surface (PS)	A short additional handbook which describes the concept and use of PS in detail.
Poly Curve System (PCS)	A short additional handbook which describes the concept and use of PCS in detail.
Shell Programming	An introduction to the LucidShape shell programming language.
Reference Manual	A description of all the data types, classes and methods which are available in LucidShape.
Technical Reference	A manual which describes several technical and optical backgrounds used in LucidShape such as e.g. different sensor types ([Cd], [lx], [lm], etc.), spectral simulation (difference between step and fit function) and more.
Tutorial Dialog Programming	A tutorial about the LucidShape dialog programming.

Appendix A. LucidShape File Types

File Format	Descriptions
.lue	Experiment files; a tree view displays the structure; a 3D view displays the geometry and UV-Data views display all light distributions.
.lus	Script files to be loaded into LucidShape such as models or also dialogs, but not the complete experiments (see .lue).
.lug	Geometry files; a 3D view displays the geometry.
.luc	Configuration files for the LucidStudio Applications.
.lub	Benchmark files for the LucidStudio Benchmark test.
.luf	Feature classes for the LucidStudio Application and geometry creation.
.lid	Light distribution; a UV-Data view displays the distribution.
.do	Script files and tasks files; a text view displays the file. You can execute the script with the execute button. The resulting printout is transferred to the message view.
.hdo	Header files; a text view displays the file. These files can be included into .lue, .lug, .do or .hdo files (like .h).
.ray	Ray files used for "ray file" light sources.

Default LucidShape File Types

File Format	Descriptions
.3dm	Rhino3D geometry files; mesh-surfaces, NURBS curves and surfaces, faces and B-reps.
.stl	Stereo lithography format with triangular patches. Binary or ASCII will be detected.
.dat	Simple Point sequences.
.iges, .igs	IGES geometry files.
.step, .stp	STEP geometry files.

Other Geometry File Types

File Format	Descriptions
.ies	IES light distributions.
.cie	CIE light distributions.
.ldt	EULUMDAT light distributions; format of European lamp maker.
.lmt	LMT goniometer format.

File Format	Descriptions
.krs	Optronik goniometer format.
.kzu	Kohzu Seiki Goniometer light intensity data.
.dis	ASAP light intensity data.
.din	ASAP light intensity data.
.csv	LMT goniometer light intensity data in Excel text.

Other Photometry File Types

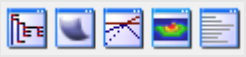
Appendix B. List of Toolbar Icons

The program shows different tool bars for each view type on focus. For a detailed description about each tool, refer to the text above in the appropriate chapter.

B.1. Common Tools

	New Document
	Open File, Add File, Open File Again
	Save File As and Save file
	Open Print Preview
	Cascade Views, Tile Views Vertically, Tile Views Horizontally
	Toggle Detach View

B.2. View Options and Manipulation Tools

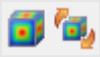
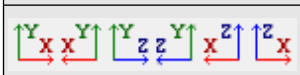
	Open TreeView, GeoView, Light Screen View, UVView, Text Edit View
	Surface Display Properties
	Reframe, Reframe Selected
	Zoom In, Zoom Out
	Fit UV Data Size to Window
	Toggle the UV data display of standard grid lines vs. polar grid lines

B.3. Simulation and Script Execution Tools

	GPU Ray Trace, Monte Carlo Ray Trace, Light Mapping Ray Trace
	Generate Random Rays, Clear Random Rays
	Interrupt Simulation or Script
	Execute Current Script File

	Clear Selection

B.4. Geometry View Tools

	Toggle Black & White, Toggle Shaded/Wireframe, Toggle Triangles/Wireframe
	Toggle Lightdata/Wireframe, Toggle Lightdata+AutoRedraw/Wireframe
	Back View, Front View, Left View, Right View, Bottom View, Top View

B.5. UV/Light Data View Tools

	Apply Most Recently Used Test Table
	Duplicate UV/LightdataView
	Filter UV/LightdataView (without or with Gradient Protection)
	Calculate the Gradient in U or V
	Navigate Through UV Data Blocks
	Add UV Dataset, Subtract UV Dataset, Deviation Between UV Datasets (A-B)/B
	Scale the UV Dataset
	Mirror the UV Dataset in U or V Direction
	Rotate the UV Dataset by 90°
	Applies the current UVViews min./max. values onto all UVViews

Appendix C. Mouse Interaction Reference

In the following, *LMB* stands for "Left Mouse Button" and *RMB* stands for "Right Mouse Button".

C.1. Geometry View

LMB	Rotates geometry within the geometry view
Double-Click LMB Over Geometry	Selects the geometry under the mouse cursor
SHIFT + LMB	Zooms in and out within the geometry view
CTRL + SHIFT + LMB	Zoom in on the rectangular region selected
CTRL + LMB	Activates the interactive ray tracers with a single ray bundle
CTRL + Double-Click LMB	Freezes the interactive ray tracer ray bundle on the surface
RMB	Activates the context menu
SHIFT + RMB	Displays surface dimensional information under the mouse cursor
CTRL + SHIFT + RMB	Displays dimensional information about the window under the mouse cursor
CTRL + RMB	Activates the interactive ray tracers with a grid of ray bundles
CTRL + Double-Click RMB	Freezes the interactive ray tracer grid of ray bundles on the surface
Scroll Wheel In & Out	Zooms in and out within the geometry view
Hold Scroll Wheel	Pans the geometry view

C.2. Tree View

LMB	Selects the tree object under the mouse cursor
Double-Click LMB Over Tree Object	Opens the edit dialog box for the tree object
CTRL + LMB	Selects multiple tree objects - one at a time
SHIFT + LMB	Selects multiple tree objects – from first selected object to the end object under the mouse cursor
RMB	Activates the context menu

C.3. Light Screen View

LMB	Displays angular coordinates under the mouse cursor
RMB	Activates the context menu
Scroll Wheel In & Out	Zooms in and out within the light screen view
Hold Scroll Wheel	Pans the light screen view

C.4. UV/Lightdata View

LMB	Displays the angular coordinates and the sensor values under the mouse cursor
SHIFT + LMB	Zooms in and out within the UV/Lightdata view
CTRL + SHIFT + LMB	Zoom in on the rectangular region selected
RMB	Activates the context menu
Scroll Wheel In & Out	Zooms in and out within the UV/Lightdata view
Hold Scroll Wheel	Pans the UV/Lightdata view

C.5. Text Edit View

LMB	Inserts the cursor into the text or selects the text
SHIFT + LMB	Selects multiple characters of test – from first cursor location to the end location under the mouse cursor
RMB	Activates the context menu

C.6. Message Box

LMB	Inserts the cursor into the text or selects the text
SHIFT + LMB	Selects multiple characters of test – from first cursor location to the end location under the mouse cursor
RMB	Activates the context menu

Appendix D. Efficacy Scale Factor and Rhino CAD Im-/Export

D.1. Efficacy Scale Factor

The efficacy factor in Section 7.2, “Emitter Materials” is used to adjust the luminous power of emitter shapes in LucidStudio to correctly reproduce the output of measured "real" light sources. A real light source (a bulb, lamp, LED) has a socket, connects, a housing, shields, shading wires... that means, there are mechanical components blocking certain spacial angles and reducing the luminous power of the bare illumination (e.g. the filament, the arc or the emitter surface). Measured data always contain the power or intensity distribution of a lamp, not the illumination only.

If a light source is covered nearly for one fifth of space by an absorbing shield (e.g. like the H4 automotive headlamp, low beam) and delivers 1000 lumen, the efficacy factor is calculated as 1/(1-loss) = 1.25. So in this case the bare illumination would have to produce about 1250 lm in order to achieve the measured output.

Make sure that the simulated light sources within LucidStudio behave like certain real light sources that have been measured. If you create an emitter shape in LucidStudio, it still differs from its counterpart in real life: Just like a mathematically perfect lambertian cylinder source always will differ from a real "lamp" (with a socket, connects, housing, shields, shading wires...). In consequence, an arbitrary 300 lm "lamp" will not equal an arbitrary 300 lm emitter shape. Therefore you would have to use the emitter shape and a more or less realistic CAD model of the real light source (the "lamp"), because the total flux can be strongly reduced by the actual geometry of a filament housing or an absorbing front cap and such (this is before being affected by the rest of your optical setup, like mirrors or lenses).

For example, an emitter coil of Lambertian type in LucidStudio is a good representation of the tungsten coil of a real halogen lamp. However, for certain setups, especially some automotive ones, you may need a light source simulation that is an even more detailed representation of an actual bulb. This is mostly the case, when precise values for illuminance or luminous intensity are expected in your experiment. This is where LucidStudio's efficacy scale can assist. Physically, efficacy is the ratio of an emitted luminous power (radiant power times sensitivity of the human eye) and the (mostly electrical) power that is used to generate it. The *efficacy scale factor* in LucidStudio is the scale factor, by which the actual luminous power of a real emitter (e.g. the tungsten coil) is *reduced* by the (real) lamp's geometry. The flux we can measure experimentally is always the flux of a lamp, not of the emitting part only. Hence, the simulated initial flux of an emitter material is *increased* in LucidStudio in order to meet the final (real) flux values we know from experimental results.

The inverse application of the efficacy scale factor allows the rescale of the overall flux of a LucidStudio emitter for exactly this purpose. For convenience, it is pre-implemented for many known automotive light sources of defined output. As a result, you only have to define the desired "real" flux for a certain light source with known geometry (e.g. 1100 lm for the H3 low beam headlamp). An efficacy scale factor has been pre-defined for this very light source and for its technical geometry. This efficacy scale will increase the emitter flux in LucidStudio to ensure that your light source delivers the flux you have chosen (e.g. 1100 lm) and thus compensates all known losses in output. It assigns the peculiarities of the complicated geometries of real illuminations to LucidStudio's easy to handle emitter materials and certain special light sources.

D.2. Materials Import/Export: Rhino CAD

When using extension "3dm", e.g. with the *Save As* command, the current geometry is saved in the Rhino CAD Format. If the file contains material shapes, the corresponding material names are also stored and are available in Rhino. It is also possible to set material names directly in Rhino to prepare for import of Rhino data in LucidShape. When now using the exact material names defined in "..\Lucid\library\LucidSMaterialLibrary.lug", the respective Shapes are created using this specific Material when loading a file into LucidStudio. The following material names are predefined and should be used both in Rhino and in LucidShape in order to use this import/export feature correctly both ways:

- *Absorber:* Absorber
- *Emitter:* Lambert Emitter, Directional Emitter
- *Reflective:* Ideal Specular (Mirror), Lambertian Reflective, Gaussian Reflective, Gaussian Retro Reflex Reflective, ABg Reflective
- *Refractive:* PC Material, PMMA Material, Glass Material (Lens), APEC Material, LEXAN Material
- *Sensor:* Lux Sensor

Index

A

Actors, 13
Approximated + Interpolated Curve
  Explicit Curve, 107
  Fitting Curve, 106
  Read Value Row, 107
Approximated + Interpolated Surface
  Create Explicit Surface, 116
  Create Fitting Surface, 115
  Create Point Grid, 112
  Create Skinned Surface, 116
  Create Surface from Point Cloud, 111
  Extrapolate Point Grid, 113
  Point Grid on Surface, 115
  Read Point Cloud, 109
  Read Point Grid, 112
  Read Value Grid, 115
  Sample Point Cloud, 110
  Sample Point Grid from Target Scene, 114
Assign Material, 187
Axis System, 119

B

BSDF, 296
BSDF Data, 227
BSDF Wizard, 224

C

CIE, 52
Clipping Plane, 120
Coil, 82
color spaces, 52
Convert to (NURBS), 73
Create Material, 187
Curve
  Approximation, 105
  Compound Parabolic Concentrator (CPC), 88
  Ellipse, 87
  Ellipse F to F, 87
  from Formula, 88
  from NURBS surface border, 90
  from Trimmed Surface Border, 90
  Helix, 88
  Hyperbola, 87
  Interpolation, 105
  Standard Curves, 86
Cylinder, 79
  from Rays, 121

D

Design Feature Manager, 130
Document
  Edit Operations, 31

E

Edit (chapter), 31
Edit operations, 37
Efficacy scale factor, 311
elevate degree of NURBS curve, 74
elevate degree of NURBS surface, 74
Ellipse, 87
Emitter, 13
extract Surfaces, 74

F

FF Geometry
  PCS reflector/lens surface, 132
File (chapter), 31
File Operations, 31

G

Geo View, 14
Geometry, 13

H

Helix, 88
Henyey-Greenstein
  Volume Scattering model, 198
Human Eye Vision Image (HEVI), 52
HWHM, 165, 195
Hyperbola, 87
Hyperboloid, 77

I

Interactive Ray-trace, 20
invert axis on NURBS surface, 74

L

Lens
  Aspherical, 96
  Aspherical, per formula, 97
  Elliptic, 103
  Free form Variable Fresnel, 102
  Fresnel, 98
  Hyperbolic, 104
  Potato Chip Lens (Outer Optic, Inner Optic), 102
  Spheric, 105
  Total Internal Reflection (TIR), 100
Light Screen View, 15

M

match (+merge) two NURBS curves
  match (+merge) two NURBS surfaces, 74
Materials, 13
Menu
  Customization, 35
Message View, 17
MF reflector/lens surface, 124
Model Tree View, 14
modify knot Vector in NURBS curve, 74
modify knot vector in NURBS surface, 74
Modify NURBS, 73
Mouse Interaction, 18

O

Object Vector, 120
Objects
  Delete, 70
  Rename, 70
  Select, 70

P

PCS
  reflector/lens surface, 132
Polyline (from Rays), 121
Principles of LucidStudio, 11

R

Ray Deviation Correction, 118
Read Point Row, 106
rebuild NURBS surface, 75
reduce degree of NURBS curve, 74
reduce degree of NURBS surface, 74
REFLET, 227
Rhinoceros
  CAD import/export, 312

S

Script integration, 20
Script View, 16
Sensors, 13
Shapes, 13
SPD, 52
Surface
  Approximation, 108
  Interpolation, 108
  Procedural Surfaces, 81
swap u- and v-axis on NURBS surface, 74
Switch Board, 130

T

Text View, 16
Tone Mapping, 52
Toolbar Icon, 307
Torus, 82
transformation, 71
  set global axis, 73

U

UV Data View, 16

V

Views, 14
  Message View, 17
  Script & Text View, 16
Volume Scattering, 197